User Manual for Beta Version of TURBO-GRD A Software System for Interactive Two-Dimensional Boundary/ Field Grid Generation, Modification, and Refinement

Yung K. Choo and John W. Slater
Lewis Research Center, Cleveland, Ohio

Todd L. Henderson
Raytheon Systems Company, Waco, Texas

Colin S. Bidwell and Donald C. Braun
Lewis Research Center, Cleveland, Ohio

Joongkee Chung
Ohio Aerospace Institute, Cleveland, Ohio

This report is a formal draft or working paper, intended to solicit comments and ideas from a technical peer group.

This report contains preliminary findings, subject to revision as analysis proceeds.

Contents

USER MANUAL FOR BETA VERSION OF TURBO-GRD: A SOFTWARE SYSTEM FOR INTERACTIVE TWO-DIMENSIONAL BOUNDARY/FIELD GRID GENERATION, MODIFICATION, AND REFINEMENT

Yung K. Choo and John W. Slater
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

Todd L. Henderson
Raytheon Systems Company
Waco, Texas 76715-4580

Colin S. Bidwell and Donald C. Braun
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

and

Joongkee Chung
Ohio Aerospace Institute
Cleveland, Ohio 44142

1.0 SUMMARY

TURBO-GRD is a software system for interactive two-dimensional boundary/field grid generation, modification, and refinement. Its features allow users to explicitly control grid quality locally and globally. The grid control can be achieved interactively by using control points that the user picks and moves on the workstation monitor or by direct stretching and refining. The techniques used in the code are the control point form of algebraic grid generation, a damped cubic spline for edge meshing, and parametric mapping between physical and computational domains. It also performs elliptic grid smoothing and free-form boundary control for boundary geometry manipulation. Internal block boundaries are constructed and shaped by using a Bézier curve. Because TURBO-GRD is a highly interactive code, users can read in an initial solution, display its solution contour in the background of the grid and control net, and exercise grid modification using the solution contour as a guide. This process can be called an interactive solution-adaptive grid generation.

TURBO-GRD with its many control features is a useful grid generation tool for unusually complex flow domain geometries. This manual describes the beta version of TURBO-GRD, which runs on SGI IRIS workstations, and uses several examples to introduce various features of the code. After the initial exposure to these sample cases, users should be able to find many creative ways to apply these features to their own unique problems. Thus, TURBO-GRD could be used as a visual interactive research tool for a nonordinary two-dimensional CFD (computational fluid dynamics) study, such as airfoil performance with ice accretion.

2.0 INTRODUCTION

2.1 What TURBO-GRD Does

TURBO-GRD is a graphical interactive software system that discretizes domain boundary edges, generates two-dimensional multiblock structured grids, and enhances two-dimensional multiblock grids. This code provides explicit local and global grid control: the user can actually grab a control point with the workstation mouse and translate it to change the grid distribution. Geometry data can be in single or multiblock format with each block boundary specified in a sequence of Cartesian coordinates (x,y). TURBO-GRD can also read and write grid and solution files in PLOT3D file format. When both the grid and solution are available, users can examine them in TURBO-GRD and can modify the grid while viewing the solution. This version runs on SGI IRIS workstations with a GL graphics library.

Developed at the NASA Lewis Research Center, TURBO-GRD uses a control point form of algebraic grid generation (refs. 1 and 2) and a parametric mapping technique (refs. 3 and 4), which make it efficient by generating the grid in the rectangular parametric domain to prevent grid tangling.

2.2 What This Manual Presents

The TURBO-GRD code is highly interactive and menu driven; therefore, its features are summarized in sections 2.1 and 2.3. and are introduced in the examples of chapters 5 and 6, which also serve as tutorials. Chapter 3 gives the input/output file structure and chapter 4, the menus. The authors expect that imaginative users can find innovative ways to employ these features when explicit grid control and refinement are needed.

2.3 Basic TURBO-GRD Features

The TURBO-GRD interactive process is presented in the schematic drawing of figure 1.
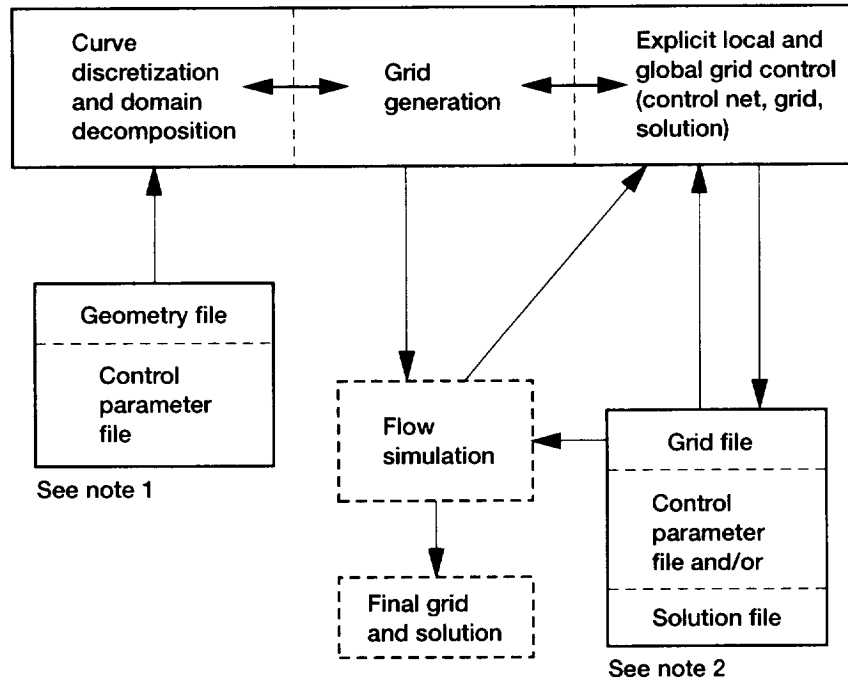
Figure 1.—TURBO-GRD features. Note 1: starting point for initial grid gener-
ation from geometry data; Note 2: starting point for grid modification when
an initial grid (and solution) is available. Flow simulation is not part of
TURBO-GRD.

Input to TURBO-GRD can be either geometry data (see note 1 of fig. 1) or grid data
alone or grid and solution data (see note 2 of fig. 1). When geometry data are given,
TURBO-GRD allows discretization of boundary curves, decomposition of the domain
when needed, generation of interior grids, and initial control net attachment. The grid can
then be modified.

When grid data are given as input, TURBO-GRD constructs a control net (CN) that gives the user explicit local or global control of the grid. The translation of a control point (CP) changes the grid within a local region around the CP. When both grid and solution are input to the code, the quality of the grid with the solution in the background can be examined and the grid structure can be controlled as needed.

2.3.1 Grid Generation Using Control Net

The control point form of algebraic grid generation provides the means to quickly generate well-structured grids. It uses a control net (CN), a sparse collection of control points (CP's) distributed over the flow domain. The shape and position of coordinate curves can be adjusted from these CP's while the grid conforms precisely to all boundaries.

Figure 2 illustrates the relationship between a CN and a grid that was generated from the CN. The structure of the grid was determined by the structure of the control net (ref. 2).
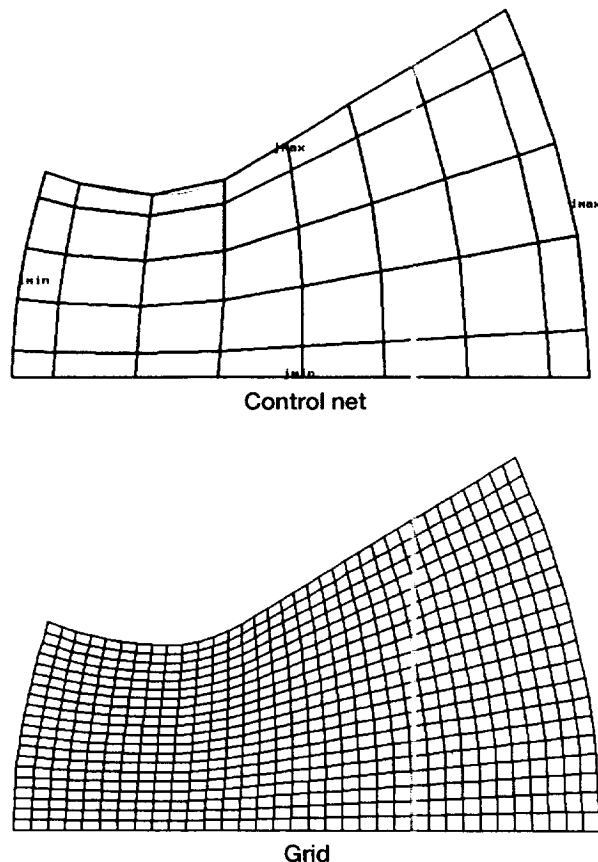
Control net

Grid

Figure 2.—Control net and grid.

2.3.2 Explicit Local Grid Control

Figure 3 illustrates the translation of a CP and its effect on the local grid distribution around the modified CP. Users can save the grid to a file at any point during the interactive process. These grid files can be used for either a flow simulation or a restart grid file for further grid control later. See example 2 in section 5.3.
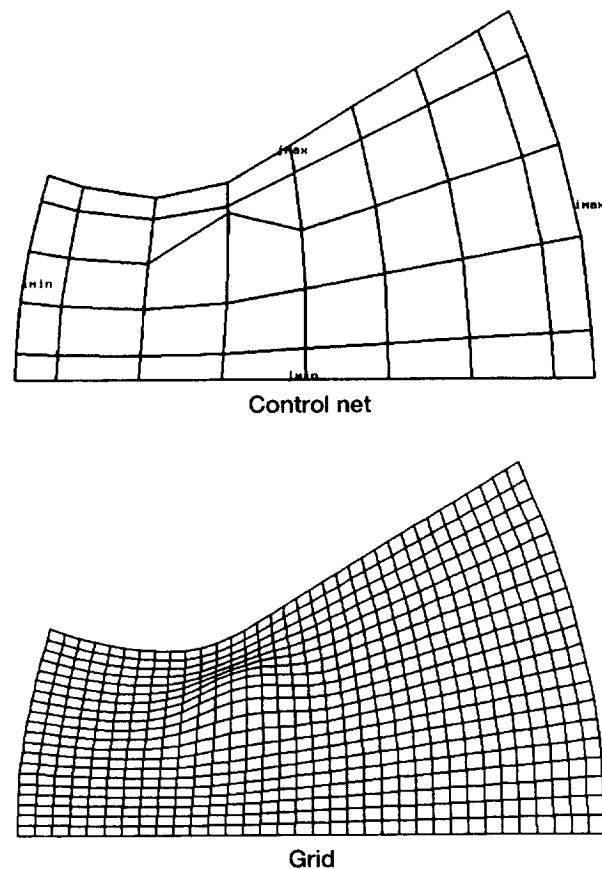
Control net

Grid

Figure 3.—Control net and grid after local modification.

2.3.3 One Step Towards Interactive CFD

The TURBO-GRD code allows users to perform grid modification and/or refinement operations with the solution contour and grid displayed in the background of the CN. This feature is useful if an initial or preliminary flow solution can be used as a guide towards a better grid or solution. Figure 4 shows an initial control net, grid, and solution. Users can display any combination of these graphical entities on the workstation monitor and operate on CN with both grid and solution in the background even though they are displayed separately in the figure.

Control net
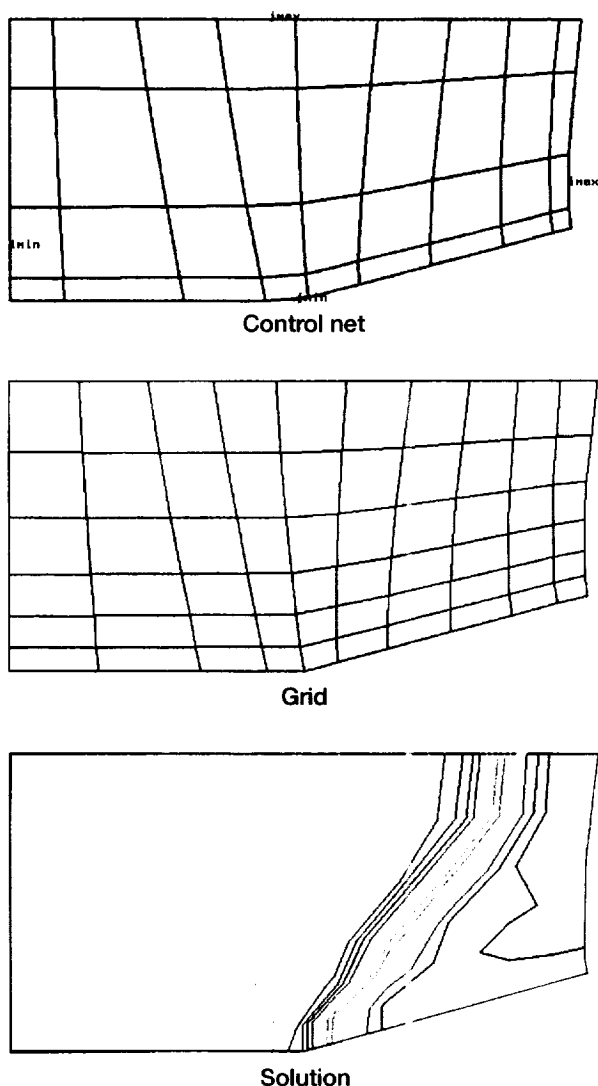
Grid

Solution

Figure 4.—Initial control net, grid, and solution.

Figure 5 shows the modified CN, the new grid generated from the modified CN, and a new solution obtained using the grid. This interactive grid adaptation will be particularly useful when flow characteristics become more complex.

Control net
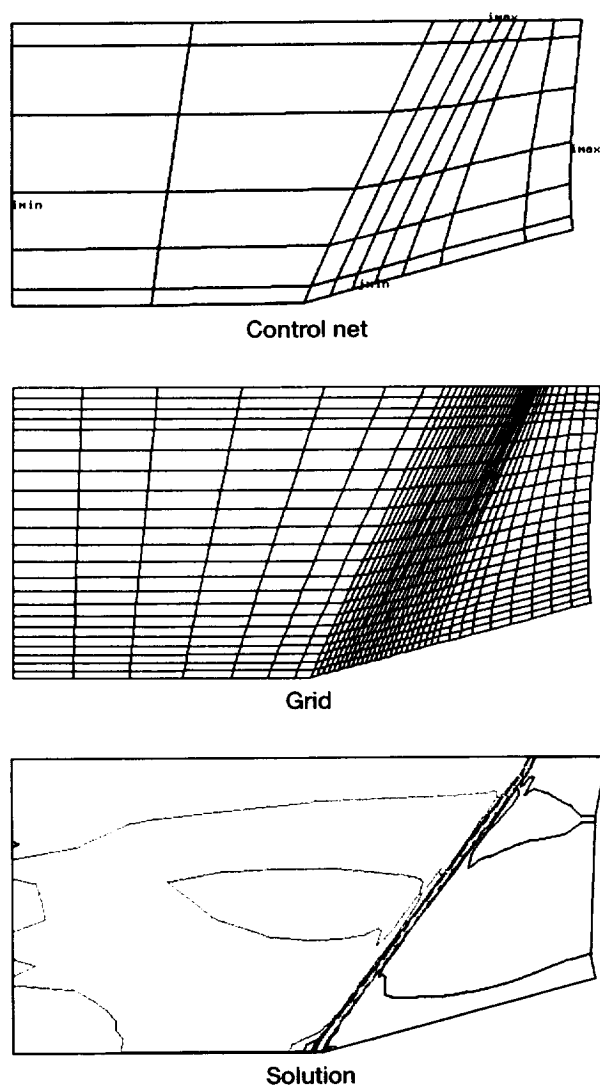
Grid

Solution

Figure 5.—Modified control net, grid, and solution.

2.3.4 Domain Decomposition and Blocked Grid Generation

With TURBO-GRD, the user can subdivide a domain interactively. Internal boundary curves are shaped by using a Bézier curve. Figure 6 shows a zoned domain for a C–H grid. Interactive steps are presented in example 4 of section 6.1.

Figure 6.—Blocked domain and boundary grid for airfoil.

The resulting multiblock grid is shown in figure 7. A sketch of the interactive steps is presented in example 6 of section 6.3.
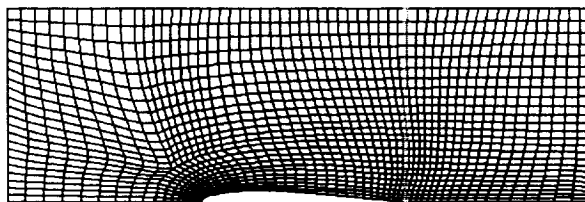
Figure 7.—C-H composite grid for airfoil.

The user can generate multiblock grids for complicated domains such as the cowl bleed passage of an inlet gridded with 11 blocks. Figures 8 and 9 show multiblock boundary and field grid, respectively. The geometry for this example was provided by the CAD-based code ICEM CFD. TURBO-GRD can take the geometry data to start with, or it can take an initial grid to modify and refine. When a block is a very small part of the entire domain, "Diagonal Zoom" and "Return to Full View" in the MODIFY VIEW menu are features essential for the interactive process.
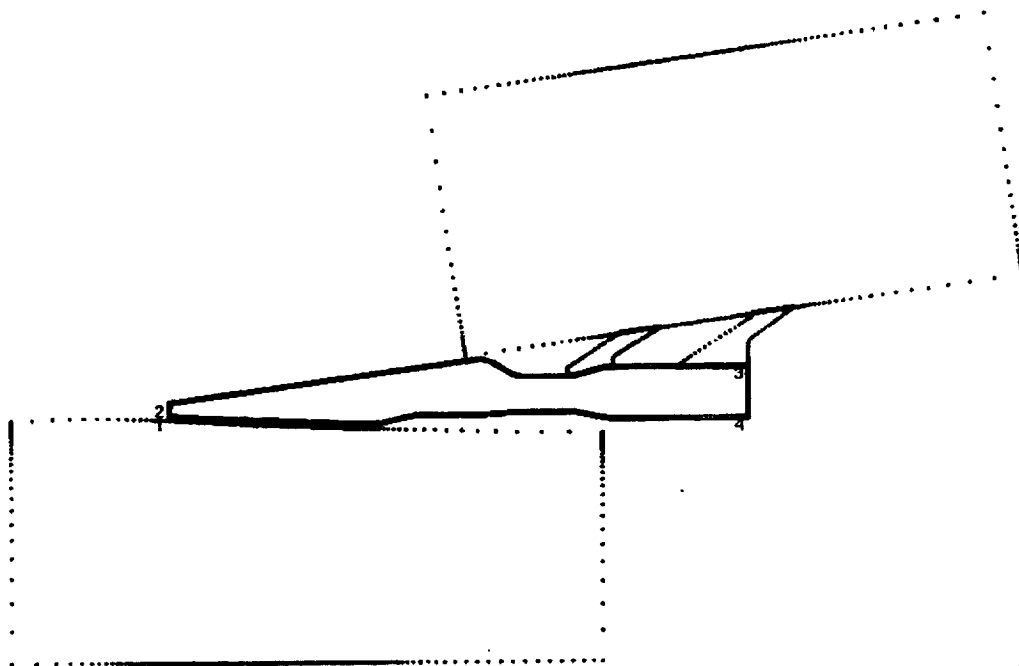
Figure 8.—Boundary grid.
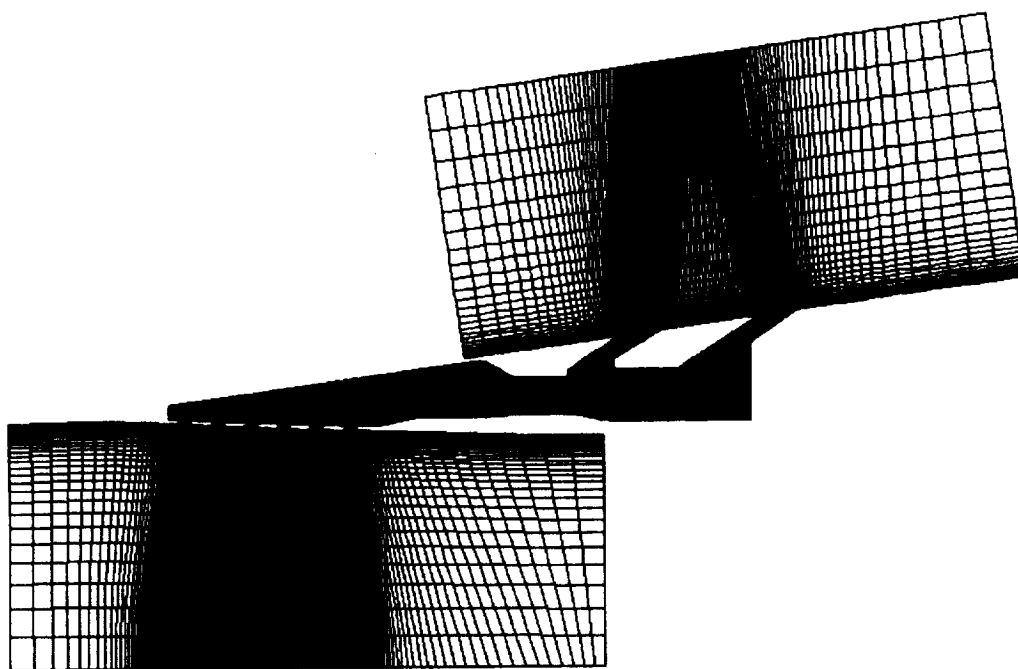
Figure 9.—Multiblock grid for cowl bleed passage.

3.0 INPUT/OUTPUT

3.1 TURBO-GRD Inputs

Several options exist to format the input and output files for the code. TURBO-GRD recognizes input data types by the input variable Data_typ_in, which is 1 for geometry data (boundary data points) and 2 for PLOT3D grid file in formatted form. Data_typ_in is specified in the CONTROL PARAMETER file to be described in section 3.1.2.

Two input files are required by TURBO-GRD: (1) CONTROL PARAMETER and (2) either GEOMETRY (for Data_typ_in = 1) or GRID (for Data_typ_in = 2 and Data_typ_in = 3). CONTROL PARAMETER files have the extension "par" (e.g., fn.par). Both GEOMETRY and GRID files have the extension "geo" (e.g., fn.geo). A solution file is an additional optional input when available. Its contour plot can be used as a guide to modify grid for an improved solution.

3.1.1 Input Files for Geometry Data

A sample four-point GEOMETRY DATA file and a CONTROL PARAMETER file for a single block are shown in tables I and II. Multiblock geometry data are presented in section 6.2.

TABLE I.—SAMPLE GEOMETRY DATA FILE (file1.geo)

Line		
1	1	
2	4	
3	0.00	0.00
4	0.00	1.00
5	1.00	1.25
6	1.00	0.25
7	0.00	0.00

Line 1: 1, a single block data
Line 2: 4, four-boundary-point data
Lines 3 to 6: x,y-coordinates of the four boundary points
Line 7: repeats the initial data on line 3

TABLE II.—SAMPLE CONTROL PARAMETER FILE (file1.par)

Line						
1	file1.par					
2	Data_typ_in	Data_typ_out				
3	1	2				
4	#blocks					
5	1					
6	imin	jmin	imax	jmax	icnmax	jcnmax
7	1	1	3	3	3	3

Line 1: heading for the input file

Line 2: heading for I/O file types

Line 3: Data_typ_in = 1 is boundary data; Data_typ_in = 2 is PLOT3D grid file in formatted form. Output file data types are specified by Data_typ_out. Data_typ_out = 2 is PLOT3D grid file in formatted form.

Line 4: heading for the number of blocks

Line 5: # block = 1 (single block)

Line 6: heading for the minimum and maximum grid indices (maximum number of control points)

Line 7: indices, the number of control points (icnmax and jcnmax) that can be interactively changed by using the submenu "Change Number of CP's" in the MODIFY GRID (Through CN) menu

How To Run TURBO-GRD With file1.geo and file1.par

USER: Type in "turbogrd" to begin the interactive process of TURBO-GRD.

TURBO-GRD: Please enter input file name!

USER: file1

TURBO-GRD: Is there a PLOT3D solution file to read? If so, what is the format of the file?
(0) No; (1) Formatted; (2) Unformatted; (3) Binary

USER: 0

TURBO-GRD: Reads in both file1.geo and file1.par. Then it displays the four data points of file1.geo on the workstation monitor and is ready to interactively carry out curve discretization (discussed and illustrated in section 5.2).

3.1.2 Input Files for Grid Data

Sample input files for a three-block grid are presented in tables III and IV. The grid data are in the PLOT3D multigrid formatted file.

TABLE III.—SAMPLE CONTROL PARAMETER FILE (file2.par)

Line						
1	file2.par					
2	Data_typ_in	Data_typ_out				
3	2	2				
4	#blocks					
5	3					
6	imin	jmin	imax	jmax	icnmax	jcnmax
7	1	1	8	15	3	5
8	9	1	18	15	3	5
9	19	1	25	15	3	5

Line 1: heading
Line 2: heading
Line 3: Data_typ_in = 2 and Data_typ_out = 2 are PLOT3D grid files in formatted form
Line 4: heading for the number of blocks
Line 5: three blocks
Line 6: heading
Lines 7 to 9: values for imin, jmin, imax, jmax, icnmax, jcnmax for blocks 1 to 3

TABLE IV.—SAMPLE GRID DATA FILE (file2.geo)

Line		
1	3	
2	8	15
3	10	15
4	7	15
5 to 4 + n1	Block 1 data	.....Grid data in PLOT3D format
	Block 2 data	.....Grid data in PLOT3D format
	Block 3 data	.....Grid data in PLOT3D format

Line 1: three-block data
Line 2: imax = 8, jmax = 15 for block 1
Line 3: imax = 10, jmax = 15 for block 2
Line 4: imax = 7, jmax = 15 for block 3
Lines 5 through 4+n1: grid coordinate data in PLOT3D format for block 1
Lines 4+n1+1 through 4+n1+n2: coordinate data in PLOT3D format for block 2
Lines 4+n1+n2+1 through 4+n1+n2+n3: coordinate data in PLOT3D format for block 3

3.1.3 Input Solution File

The TURBO-GRD code reads in either a formatted, unformatted, or SGI binary PLOT3D multiblock solution file. The solution contour can then be used as a guide for explicit grid control. After the grid is changed, the solution is reinterpolated on the new grid. This solution file and new grid can then be saved.

3.2 TURBO-GRD Outputs

The following output files are created by TURBO-GRD: CONTROL PARAMETER (xxx.par) and GRID (xxx.geo). The xxx.par file has the same format as that of the corresponding input parameter file described in section 3.1.2 with Data_typ_out = 2 for grid. The xxx.geo file is written in PLOT3D format. These output files can be used as input files whenever further grid modification is desired in a later TURBO-GRD session. The GRID file (xxx.geo) is created by the program fragment shown in table V:

TABLE V.—WRITE STATEMENTS FOR OUTPUT FILE GRID

```
write(38,*)iblk
do n = 1, iblk
write(38,*) igmax(n), jgmax(n)
enddo
do n = 1, iblk
write(38,*) (x(i, j, 1, n), i = 1, igmax(n)), j = 1, jgmax(n))
write(38,*) (y(i, j, 2, n), i = 1, igmax(n)), j = 1, jgmax(n))
enddo
```

As indicated earlier, TURBO-GRD can use an existing solution as a guide to modify grid for a better solution. After such grid modification, TURBO-GRD interpolates the solution onto the new grid. The "Update Solution" in the MODIFY GRID (Through CN or Direct) menu is used for the interpolation and the "Save Solution to a File" in the HOME menu is used to create the updated solution file Solout.

Recreating the grid using a history file is not practical because TURBO-GRD is a highly interactive code that has an explicit local control capability. Instead, users can save grid files at key stages along the interactive process.

4.0 TURBO-GRD MENUS

An interactive code, TURBO-GRD uses mouse commands and menus to perform the discretization of boundary curves, the blocking of a domain, the generation of an algebraic grid, and the overall quick modification of the grid structure through control point movement. Direct grid refinement is also available for final fine tuning. The TURBO-GRD menu structure resides in a separate file called *menulist*. In table VI, the main menus as they appear on the monitor are in the left-hand column. In the right-hand column are explanations. The submenus allow the user to go from one main menu to another.

TABLE VI.—MAIN MENUS

HOME	Contains all other main menus; also contains submenus to save files, to quit from TURBO-GRD run, to switch blocks, to restore control net and grid, to choose active subblock, to toggle displays
MODIFY VIEW	Contains submenus to modify display of objects on screen, such as return to full view and diagonal zoom, to toggle displays, and to set functions
GENERATE BNDRY/FIELD GRID	Contains submenus to generate boundary grids, to interactively decompose domains, to discretize subcurves, to subdivide blocks
MODIFY GRID (Through CN)	Contains submenus to modify initial or existing grids by using control net, to update solutions, to change number of control points, to display point coordinates, to make control net orthogonal to boundaries
MODIFY GRID (Direct)	Contains submenus to modify initial or existing grids by using direct stretching and refining, to perform Laplace and elliptic smoothing

4.1 HOME Menu

The HOME menu contains all other main menus but also includes its own submenus to save grid files, to save solution files, to switch blocks in a multiblock operation, to restore most recently computed control net (CN)/grid, to save PARAMETER/GRID files, to save RGB files, and to quit the TURBO-GRD session. The HOME menu appears on the monitor as shown in table VII:

TABLE VII.—HOME MENU

```
*HOME*

MODIFY VIEW
GENERATE BNDRY/FIELD GRID
MODIFY GRID (Through CN)
MODIFY GRID (Direct)
QUIT
Switch Blocks
Choose Active Subregion
Display Point Coordinates
Toggle Display of Grid
Toggle Display of CN
Toggle Display of Solution
Save RGB File
Save Grid to a File
Save Solution to a File
```

4.2 MODIFY VIEW Menu

This menu contain submenus to modify the display of objects on the monitor. Table VIII shows the menu and entries as they appear on the monitor:

TABLE VIII.— MODIFY VIEW MENU

```
*MODIFY VIEW*

Return
Diagonal Zoom
Gradual Zoom and Move
Return to Full View
Choose Active Subregion
Toggle Display of Grid
Toggle Display of CN
Toggle Display of Solution
Set Function to Density
Set Function to X-momentum
Set Function to Y-momentum
Set Function to Tot Energy
Set Function to Pressure
Set Function to Mach Number
Set Function to X-velocity
Set Function to Y-velocity
```

4.3 GENERATE BNDRY/FIELD GRID Menu

This menu contains submenus to discretize curves. One can interactively choose a curve segment, assign the number of points, stretch them, and set corners that correspond to the corners of the topologically rectangular parametric domain. When the boundary grid is completed and the corners are set, the "Construct Initial CN and Grid" submenu should be selected to generate the initial grid and to attach a control net to it. TURBO-GRD automatically brings the interactive process to HOME. The GENERATE BNDRY/FIELD GRID menu appears on the workstation monitor as shown in table IX.

TABLE IX.— GENERATE BNDRY/FIELD GRID MENU

```
*GENERATE BNDRY/FIELD GRID*

MODIFY VIEW
HOME
Choose Subcurve
Discretize Subcurve
Switch Blocks
Save Boundary Grid
Free Form Control
Construct Initial CN and GRID (pulldown menu)
Display Point Coordinates
Select Corners
Subdivide Block
Copy Subcurve Distribution
Apply Subcurve Distribution
Save RGB File
QUIT
```

4.4 MODIFY GRID (Through CN) Menu

The MODIFY GRID (Through CN) menu contains submenus to modify the control net structure. "Recalculate Grid" computes a new grid using the modified control net. When grid modification is completed, the solution, if read in, should be updated for the new grid. This update can be accomplished by selecting "Update Solution." The MODIFY GRID (Through CN) menu appears on the monitor as shown in table X and the options available in the pulldown menu "Make CN Orthogonal to BNDRY" are shown in table XI.

TABLE X.—MODIFY GRID (Through CN) MENU

```
*MODIFY GRID (Through CN)*

MODIFY VIEW
HOME
Choose and Move CP
Change Number of CP's
Display Point Coordinates
Stretch CN
Make CN Orthogonal to Boundary (pulldown menu)
Toggle Display of Grid
Toggle Display of CN
Toggle Display of Solution
Update Solution
Recalculate Grid
```

TABLE XI.— MAKE CN ORTHOGONAL
TO BOUNDARY MENU

```
*Make CN Orthogonal to BNDRY*

Normalize Top (2-CP levels)
Normalize Top (1-CP level)
Normalize Bottom (2-CP levels)
Normalize Bottom (1-CP level)
Normalize Left and Right
```

4.5 MODIFY GRID (Direct) Menu

The grid can also be modified directly without using the control net. The MODIFY GRID (Direct) menu appears on the monitor as shown in table XII.

TABLE XII.— MODIFY GRID (Direct)
MENU

```
*MODIFY GRID (Direct)*

MODIFY VIEW
HOME
Display Point Coordinates
Stretch Grid
Refine Grid
Choose Active Subregion
LaPlace Smoothing
Elliptic Smoothing
Update Solution
```

5.0 TURBO-GRD FEATURES THROUGH SINGLE-BLOCK EXAMPLES

5.1 Menus To Operate TURBO-GRD

To select a menu item, move the cursor to the desired item and press the right mouse key (RM-key). Other operations under an entry can be executed using the left mouse key (LM-key) or the middle mouse key (MM-key) as instructed on the monitor. This differs from widely used conventions.

5.2 Example 1: Curve Discretization and Initial Grid Generation

This example demonstrates the use of TURBO-GRD features for (1) boundary grid generation starting from given geometry data and (2) initial grid and/or control net (CN) generation.

The input files of example 1 are file3.geo and file3.par. The file3.geo provides converging-diverging nozzle geometry data for 14 boundary points, as shown in table XIII. The file3.par file shown in table XIV provides the parameter values.

TABLE XIII.— file3.geo

Line		
1	1	
2	14	
3	1.014	0.000
4	1.032	.274
5	1.084	.552
6	1.138	.749
7	1.300	.700
8	1.466	.673
9	1.635	.670
10	1.795	.724
11	1.940	.812
12	2.084	.902
13	2.807	.346
14	2.951	.973
15	3.074	.476
16	3.117	.000
17	1.014	.000

Line 1: geometry data for one block
Line 2: 14 data points
Lines 3 through 16: x,y-values of the 14 data points
Line 17: repeats the initial data on line 17

TABLE XIV.—file3.par

file3.par

Data_typ_in	Data_typ_out				
1	2				
#blocks					
1					
imin	jmin	imax	jmax	icnmax	jcnmax
1	1	29	15	11	5

5.2.1 Boundary Grid Generation

How To Run TURBO-GRD With file3.geo and file3.par.

USER: Type in "turbogrd."

TURBO-GRD: Please enter the input file name!

USER: file3

TURBO-GRD: Is there a PLOT3D solution file to read? If so, what is the format of the file?

(0) No; (1) Formatted; (2) Unformatted; (3) Binary

USER: 0

TURBO-GRD: Reads in both file3.geo and file3.par and displays the geometry as shown in figure 10. When Data_typ = 1, the code puts the GENERATE BNDRY/FIELD GRID menu on the monitor.

How To Choose Subcurve To Be Discretized

USER: Select "Choose Subcurve." Point the cursor at one point at the end of the curve and press the LM-key; then move the cursor to the other end point of the subcurve and press the MM-key.

TURBO-GRD: Highlights the points in red on the selected subcurve that represents the nozzle surface of this example (fig. 10). The code displays the number of points on the selected subcurve in the menu box on the right-hand side of the monitor (not shown here).

The left or right arrow key can be used to select the second point in increments of one grid point per stroke.
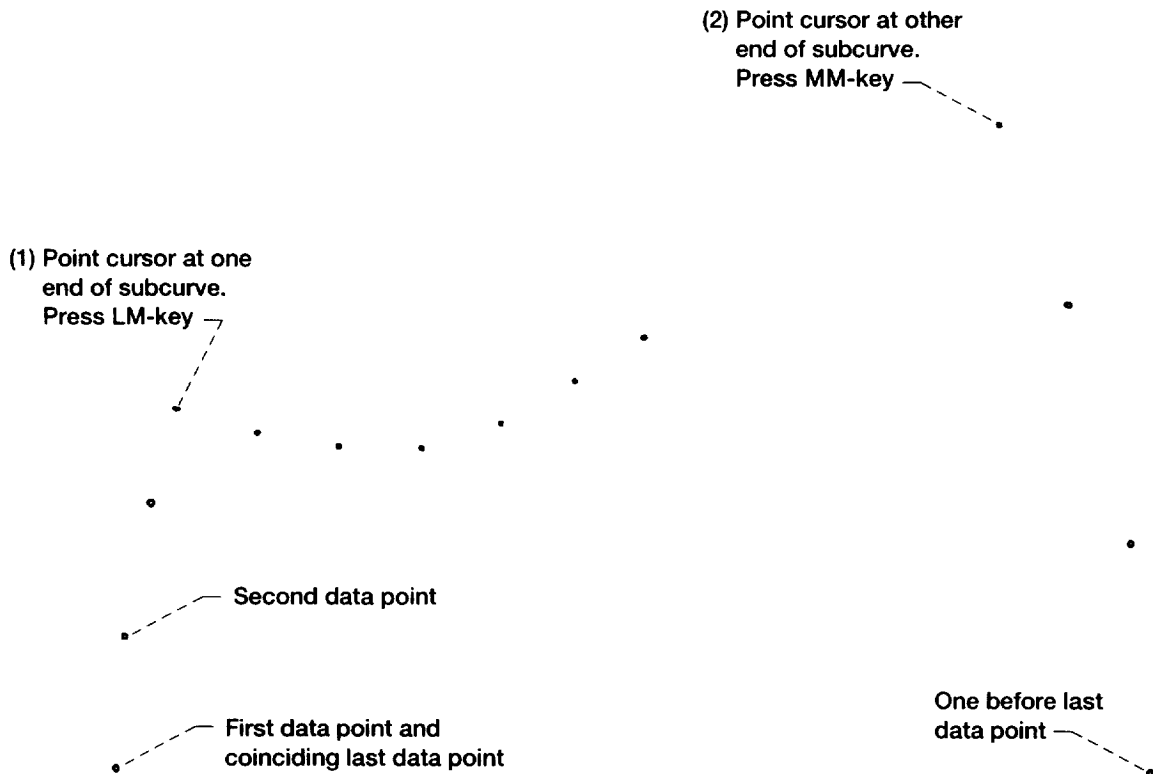
(2) Point cursor at other end of subcurve. Press MM-key

(1) Point cursor at one end of subcurve. Press LM-key

Second data point

One before last data point

First data point and coinciding last data point

Figure 10.—Input geometry data and selection of subcurve.

How To Discretize Selected Subcurve

The variable "unif" below is a parameter to obtain a uniform grid distribution. If unif = 0, then any existing grid distribution is maintained; if unif = 10, the grid distribution becomes uniform. Grid distribution becomes more uniform as its value increases from 0 towards 10.

USER: Select "Discretize Subcurve."

TURBO-GRD: Displays unif = 0, crv = 0, pnts = 8 in the menu box on the right-hand side of the monitor. The unif = 0 means that the code will maintain the current distribution when the user changes the number of points; crv = 0 means that the curvature will not affect the point distribution; and pnts indicates the number of points on the chosen subcurve.

The parameter "crv" is intended to have a denser grid where the curvature is high. Since crv is not fully tested, it is recommended that the user try a gradual increase of its value from zero.

USER: Change the pnts value from 8 to 35 and the unif value from 0 to 10 for a uniform distribution. Then select "Apply Subcurve Distribution" from the GENERATE BNDRY/FIELD GRID menu.

TURBO-GRD discretizes the curve in a uniform distribution with the 35 points, as shown in figure 11. The user may try a nonzero crv value to see how it affects the points distribution. As the crv value increases, more points congregate around the nozzle throat where the curvature is the highest.
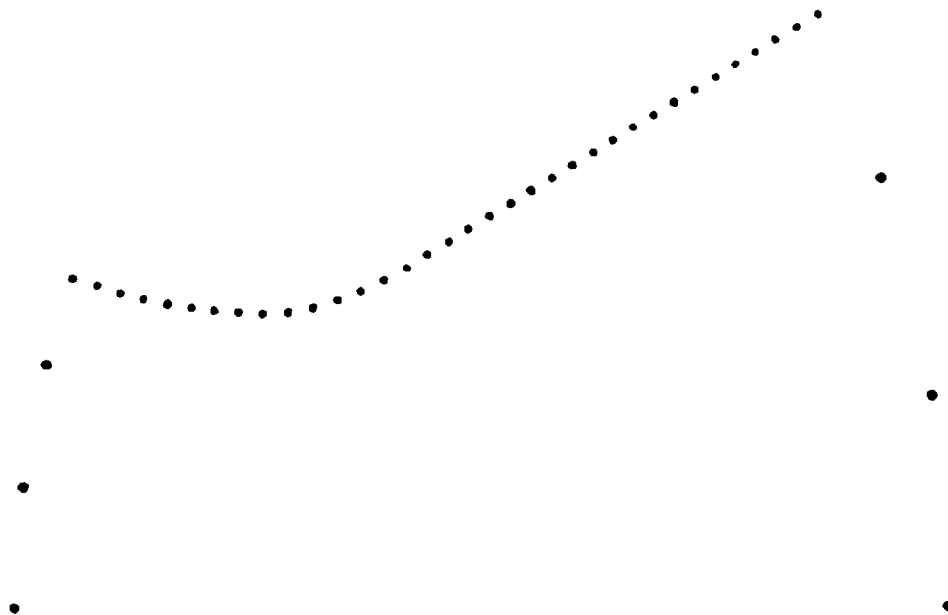

Figure 11.—Uniform grid for nozzle surface.

How To Select Bottom Boundary

The bottom boundary of figure 11 consists of two data points, (i.e., the last data point that coincides with the first and the one before the last). Even though the user intends to select the bottom subcurve that directly connects these two points, TURBO-GRD may instead select the curve that connects the first data point and the one before the last data point through the left/top/right edges. Figure 12 explains the procedure for selecting the bottom edge.
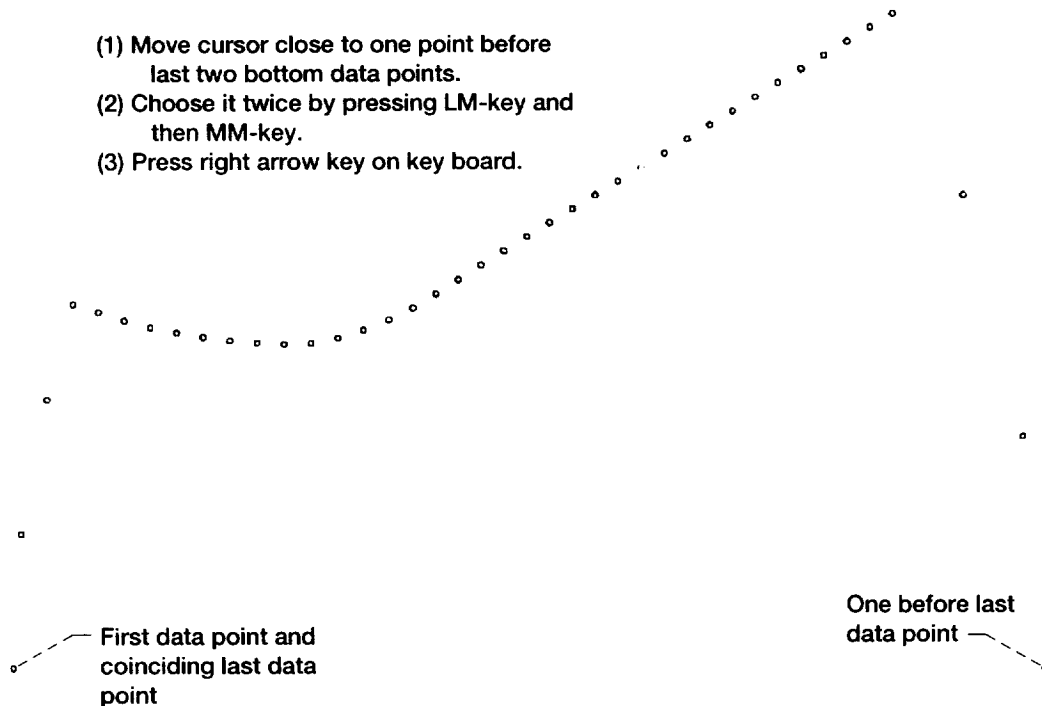
(1) Move cursor close to one point before last two bottom data points.
(2) Choose it twice by pressing LM-key and then MM-key.
(3) Press right arrow key on key board.

One before last data point

First data point and coinciding last data point

Figure 12.—Selection of two-point subcurve at bottom to directly connect last two data points at the bottom.

After the last two bottom points are selected, the discretization procedure is the same as that for the top boundary of figure 12. The bottom edge is also discretized with 35 points and the left and right edges with 21 points.
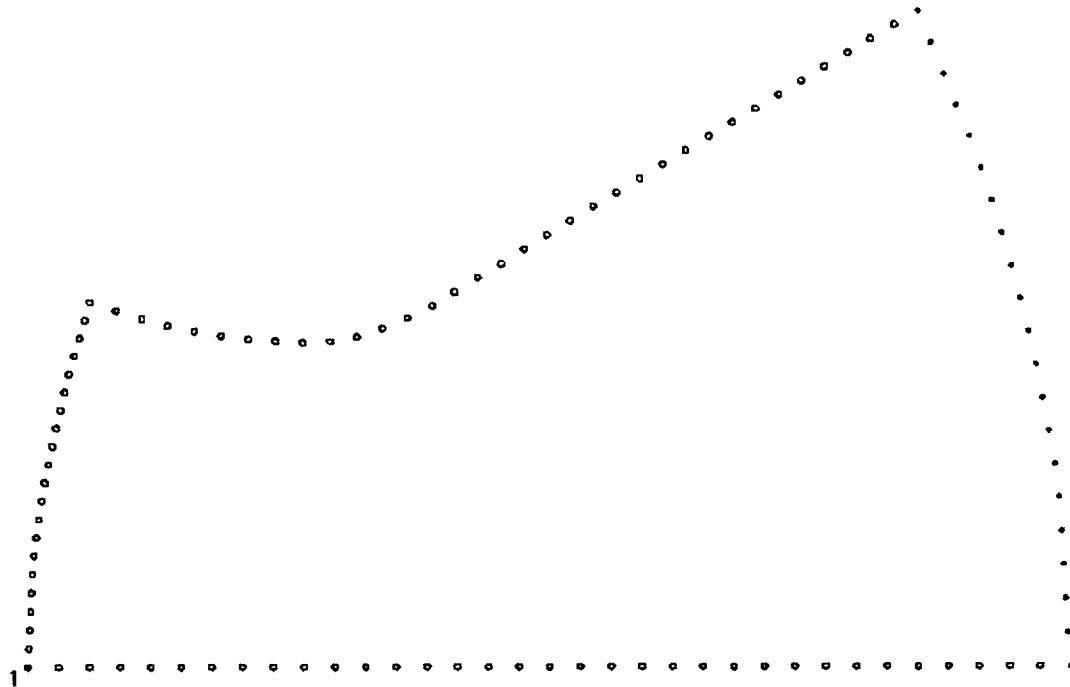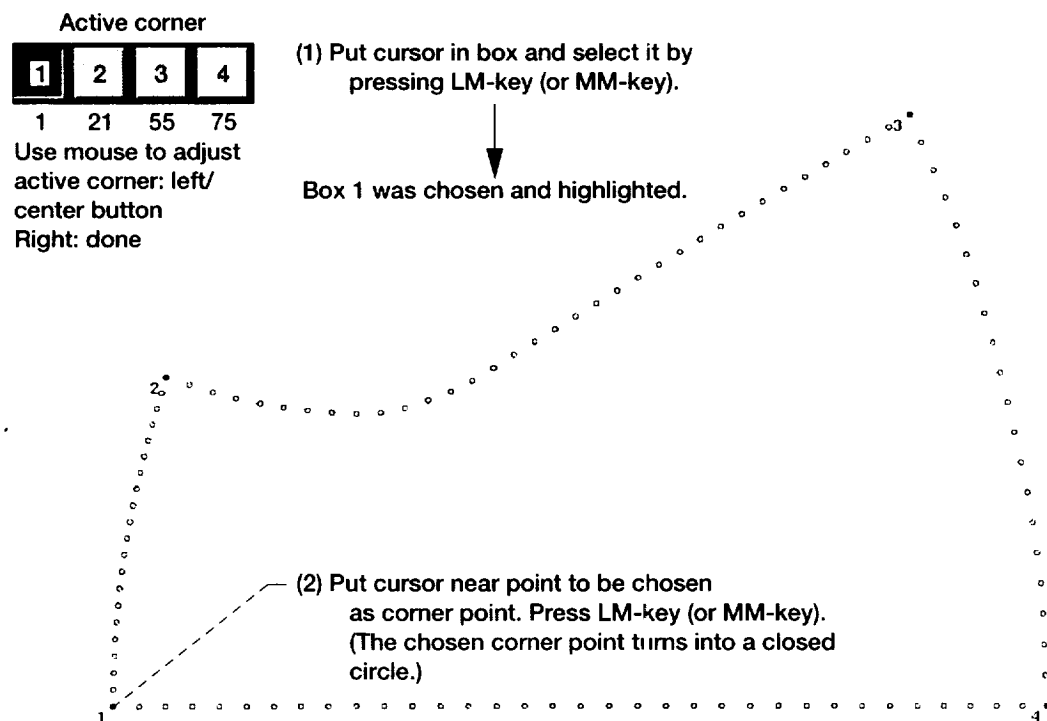
Figure 13 shows a completed boundary grid.

Figure 13.—Uniform boundary grid.

How To Set Corners Before Generating Initial Grid

USER: Choose "Select Corners."

TURBO-GRD: Displays the active corner boxes at the left upper corner of the monitor, as shown in figure 14

USER: Highlight one of the four corner boxes with the cursor and LM-key (or MM-key). Then move the cursor to the boundary point to be set as a corner point and press the LM-key (or MM-key). Repeat the process until all four corner points are selected. The corner point numbering should monotonically increase or decrease in a clockwise direction. In figure 14, the left and right boundaries have 21 points and the bottom and top, 35 points. The number of points is displayed in the lower part of the menu box. Now TURBO-GRD is ready to generate an initial grid and construct an initial control net.

Active corner

1	2	3	4

1   21   55   75
Use mouse to adjust
active corner: left/
center button
Right: done

(1) Put cursor in box and select it by
    pressing LM-key (or MM-key).

Box 1 was chosen and highlighted.

(2) Put cursor near point to be chosen
    as corner point. Press LM-key (or MM-key).
    (The chosen corner point turns into a closed
    circle.)

Figure 14.—Completed boundary grid with corners specified.

After the four corners are selected, return to GENERATE BNDRY/FIELD GRID menu. TURBO-GRD displays the number of grid points on each edge.

NOTE: Verify that the numbers of the opposing boundary edges are the same.

5.2.2 Initial Grid Generation and Control Net Attachment

Now TURBO-GRD is ready to generate the initial grid and the control net.

USER: From the menu GENERATE BNDRY/FIELD GRID, select "Construct Initial CN and GRID," which is a pulldown menu that has two entries: "Current Block" and "All Blocks." Either one will generate the grid because this is a single-block case.

TURBO-GRD: Generates the initial grid and control net and displays them overlapping each other, although they are displayed separately in figure 15

Control net
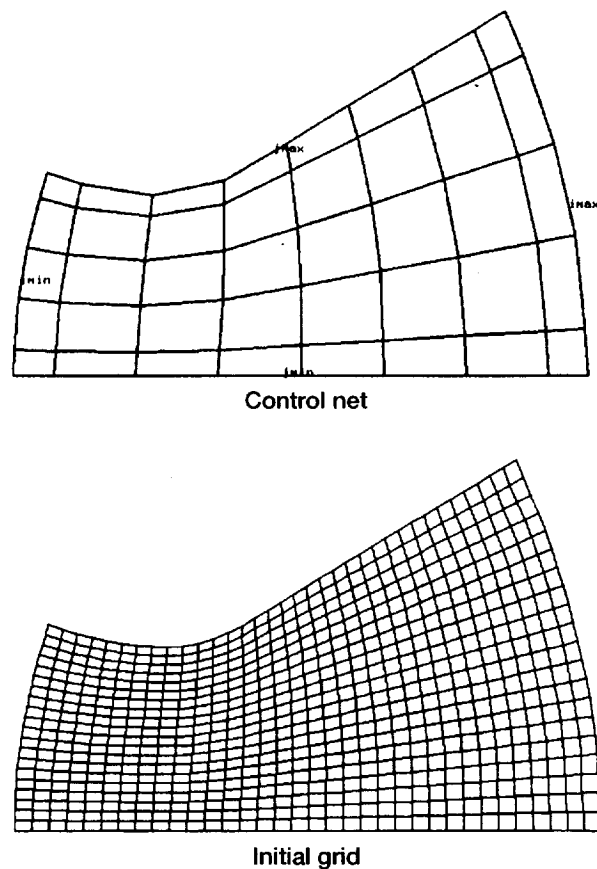
Initial grid

Figure 15.—Control net and initial grid.

How To Save Grid Before Quitting

USER: Select the HOME menu and then the submenu "Save Grid to a File."

TURBO-GRD: Invites user to specify own file name

USER: Provide file name. Choose QUIT to exit program.

TURBO-GRD: Saves files

5.3 Example 2: Explicit Grid Modification Using Control Points and Grid Stretching

Example 1 illustrated how to select a curve segment to discretize, to set corners, and to generate the initial grid and control net. This example will illustrate TURBO-GRD features that provide explicit local and global grid modification by using control points. Begin with the given boundary grid shown in figure 16.

How To Use Control Points To Modify Grid

USER: Type "turbogrd."

TURBO-GRD: Please enter input file name!

USER: file4

TURBO-GRD: Is there a PLOT3D solution file to read? If so, what is the format of the file?
(0) No; (1) Formatted; (2) Unformatted; (3) Binary

USER: 0

TURBO-GRD: Displays boundary grid as shown in figure 16
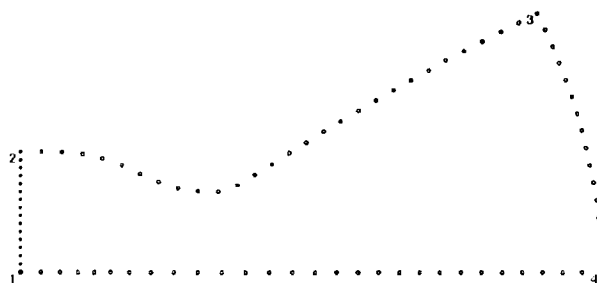
Figure 16.—Given boundary grid for example 2.

USER: From the menu GENERATE BNDRY/FIELD GRID, select "Construct Initial CN
and Grid."

TURBO-GRD: Generates initial grid and constructs initial control net by attachment. The
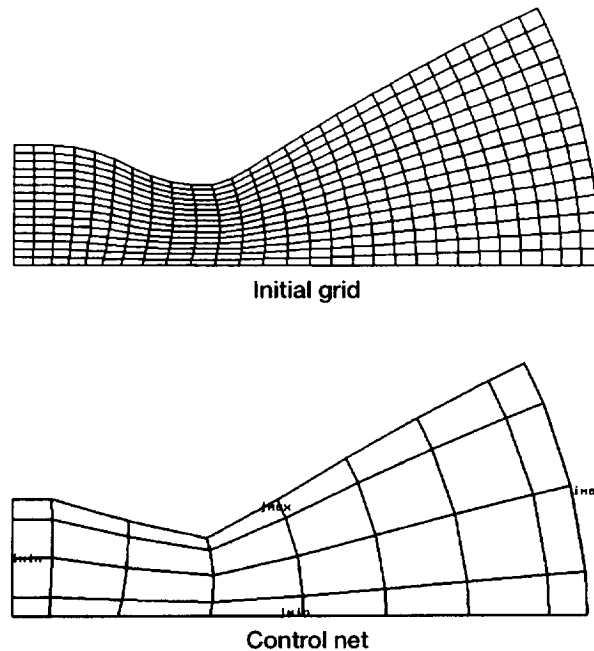resulting grid and control net are displayed in figure 17.

Initial grid

Control net

Figure 17.—Initial grid and control net.

With TURBO-GRD, the user can display both grid and control net, grid alone, or control
net alone by selecting "Toggle Display of Grid" or "Toggle Display of CN," available in
the HOME, MODIFY VIEW, and MODIFY GRID (Through CN) menus. In actual
operation, the CN is shown on top of the grid in distinct colors.

How To Examine Grid Structure and Modify It Using Control Points (CP's)

A unique capability of TURBO-GRD is the explicit local grid control. This example illustrates some of the essential control features. The initial grid in figure 17 is not orthogonal to the top and bottom surfaces and has uniform distribution. For illustration, change the grid structure by changing the structure of the control net. Specifically, the nonorthogonal grid will be modified so that it becomes orthogonal to the top and bottom surfaces.

How To Make Control Net Orthogonal To Boundaries

USER: Select the MODIFY GRID (Through CN) menu.

TURBO-GRD: Displays the menu

USER: Use the pulldown menu "Make CN Orthog to BNDRY." Choose "Normalize Top (2-CP levels)."

TURBO-GRD: Changes the control net from that shown in figure 17 to that shown in figure 18.
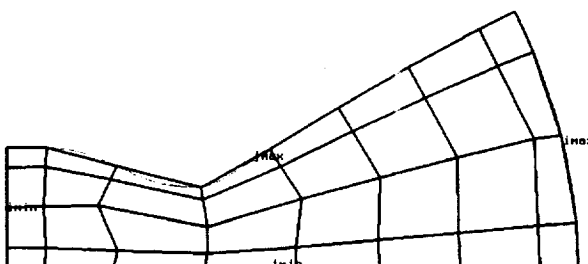
Figure 18.—Control net orthogonal to top surface.

This process can be considered a semiglobal modification because in one menu selection, several control lines are made orthogonal to the top surface. At this point, the user should tell TURBO-GRD to recalculate the grid using the modified control net in figure 18.

USER: From the MODIFY GRID (Through CN) menu, choose "Recalculate Grid."

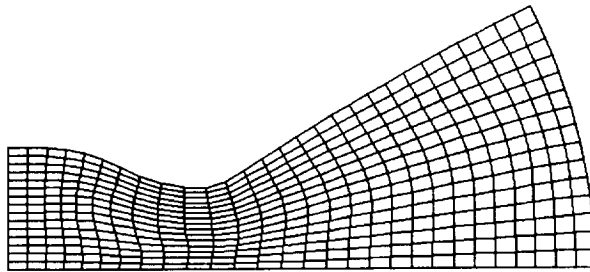TURBO-GRD: Computes new grid based on the modified control net and displays it as shown in figure 19

Figure 19.—Modified grid orthogonal to top surface.

A comparison of this modified grid with the initial grid in figure 17 clearly shows that the grid structure has been changed from one that is nonorthogonal to one that is orthogonal to the top surface.

How To Pick a Control Point and Translate From One Location to Another

Users can choose any single control point (CP) and move it to a new location. This feature can be used for both local and fine grid control (refs. 3 and 4).

USER: Select "Choose and Move CP" from the MODIFY GRID (Through CN) menu. Move the cursor to any desired CP, select it by pressing the LM-key, then move the mouse to relocate the selected CP while pressing it down (e.g., select the CP at point A and move it to point B). The LM-key is released at the desired CP location.

TURBO-GRD: Moves the CP from location A to B as shown in figure 20

Location B → ∼— Location A
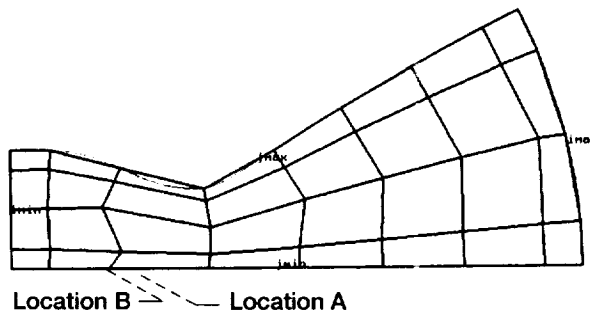
Figure 20.—Moving a control point from one location to another.

Another way to achieve it is to move the cursor to location B and then press the LM-key. The CP will be drawn from location A to B.

How To Make Control Net Orthogonal to Top or Bottom Boundary

USER: Select the MODIFY GRID (Through CN) menu.

TURBO-GRD: Displays the menu

USER: Use the pulldown menu "Make CN Orthogonal to BNDRY" to select "Normalize Bottom (1-CP level)" (see fig. 21).

How To Make Local Modification

USER: For small changes in CP locations, a simpler process is to move the cursor to a desired location; a CP will be drawn there by pressing the LM-key.

NOTE: The CP closest to the cursor location will be drawn.

TURBO-GRD: Translates the CP to desired location

USER: Repeat this operation for other CP's.
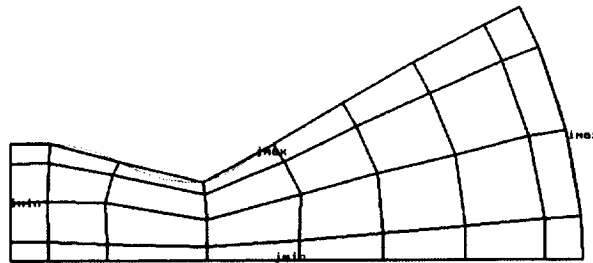
Figure 21.—Control net orthogonal to top and bottom boundaries.

USER: Select the submenu "Recalculate GRID" to obtain the new grid in figure 22.

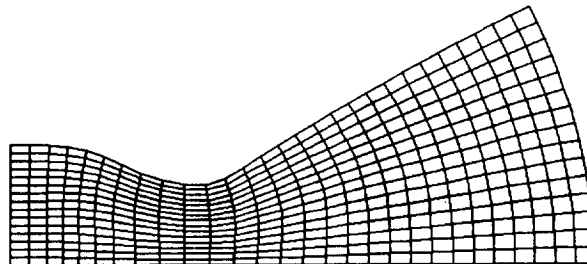TURBO-GRD: Displays new modified grid shown in figure 22

Figure 22.—Orthogonal grid.

5.4 Example 3: Grid Stretching and Refinement

5.4.1 Grid Stretching

Grid stretching can be accomplished in two ways: (1) by CN stretching and (2) directly by using stretching functions such as the hyperbolic tangent. For inviscid computation, either approach can be used but for finer grids for viscous computation, the second approach in combination with refinement will provide the desired results.

How To Stretch With Control Net (CN)

USER: Select the HOME menu and then the menu MODIFY GRID (Through CN).

TURBO-GRD: Displays MODIFY GRID (Through CN)

USER: Select "Stretch CN."

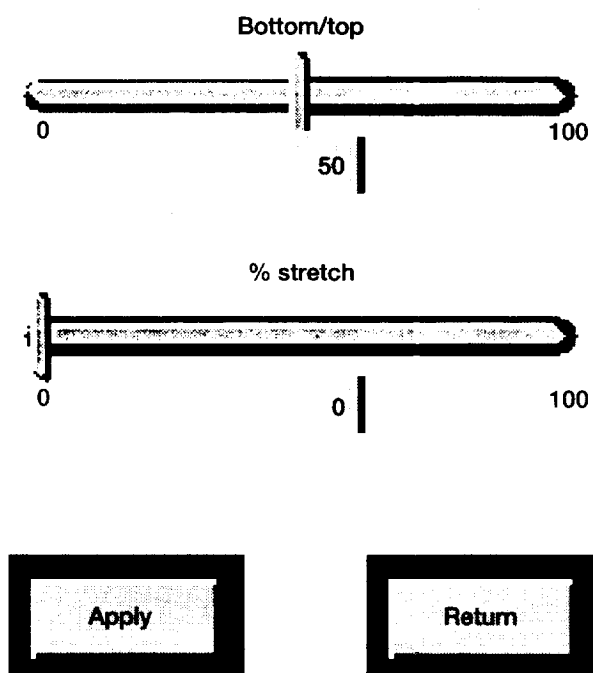TURBO-GRD Displays the menu box shown in figure 23

Figure 23.—Menu box for control net stretching.

USER: Put the cursor where desired and press the RM-key to make the bottom/top and % stretch settings. If the grid is to be stretched equally towards both top and bottom boundaries, keep the bottom/top setting at 50 (default). If the stretching is to have fine grid near the top boundary with no grid density at the bottom, set the bottom/top near 100. Similarly, if fine grid at the bottom is desired, set bottom/top near 0.

When the % stretch is reset, the instant stretching of CN occurs on the workstation monitor. Resettings can be repeated until the CN stretching is visually acceptable.

Return to MODIFY GRID (Through CN) and select "Recalculate Grid."

TURBO-GRD: With bottom/top = 50 and % stretch = 50, the control net in figure 21 is stretched to that seen in figure 24, which in turn generates the stretched grid in figure 25.
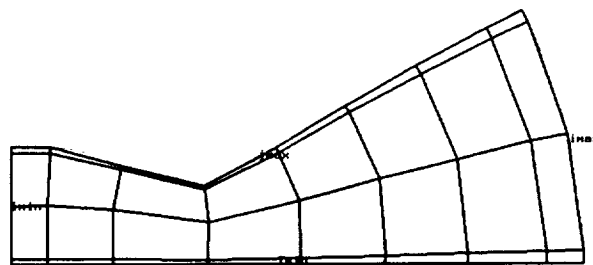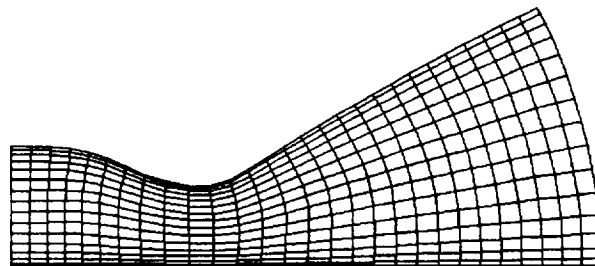

Figure 24.—Stretched control net.

Figure 25.—Grid obtained from stretched control net.

How To Stretch Grid Directly

USER: Select the MODIFY GRID (Direct) menu and then "Stretch Grid."

TURBO-GRD: Displays menu for direct geometric grid stretching shown in figure 26

USER: Select menu options by pointing the cursor to those desired and by pressing the RM-key. For numerical values, use the keyboard numeric keys.

Figure 26.—Menu for direct geometric grid stretching.

USER: After all the options are selected and/or specified, choose the "Apply" box.

TURBO-GRD: Performs geometric stretching and displays the resulting grid shown in figure 27
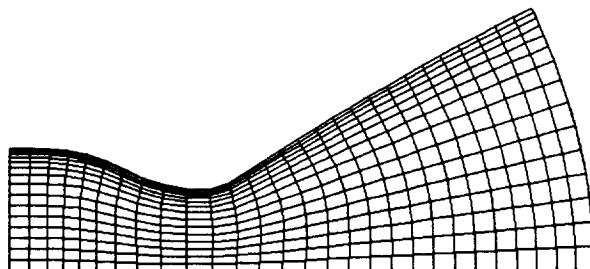
Figure 27.—Grid stretched to top.

USER: Change both top and bottom to 1250/1000.

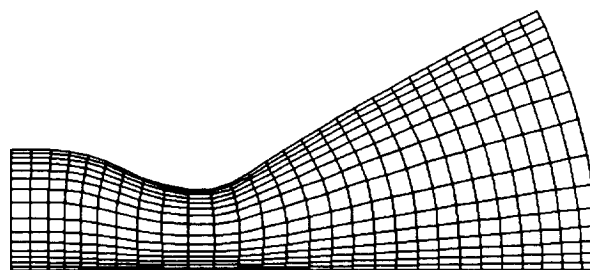TURBO-GRD: Produces the stretched grid shown in figure 28

Figure 28.—Grid stretched to top and bottom.

How To Stretch Grid Directly With Mesh Size Specified

The user can specify the size of the first mesh from the boundary. With this value, TURBO-GRD stretches grid iteratively by using the hyperbolic tangent function. In the flow domain shown in figure 29, the y-dimension between P1 and P2 is 0.670655, since y at P1 is 0.670655 and y at P2 is 0.

P1

x = 1.62741E + 00
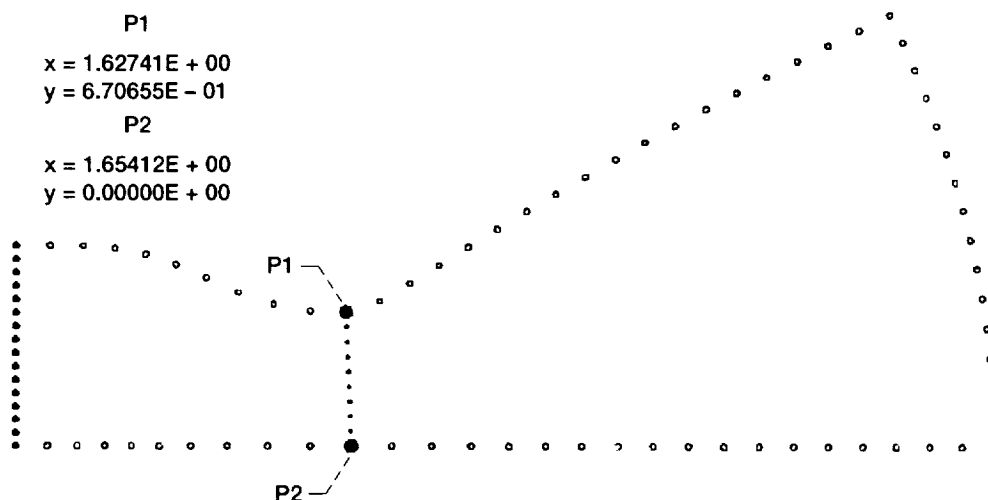y = 6.70655E – 01

P2

x = 1.65412E + 00
y = 0.00000E + 00

Figure 29.—Aid to explain "absolute" and "relative" stretching with figures 30 and 31.

For example, a user wants the first mesh size to be 0.006 at the top and 0.67 at the bottom. Set the "Direct Stretching of the grid" menu window by using the RM-key and the keyboard numeric keys as shown in figure 30.

Direct stretching of the grid

Geometric

Relative

TANH

Absolute

Top   6   /   1000

Bottom   670   /   1000

Apply   Return

Figure 30.—Interactive grid stretching menu box.

USER: Select the "Apply" box.

TURBO-GRD: Stretches grid and displays it as shown in figure 31

> NOTE: Any grid finer than 0.001 should be produced by the refinement feature
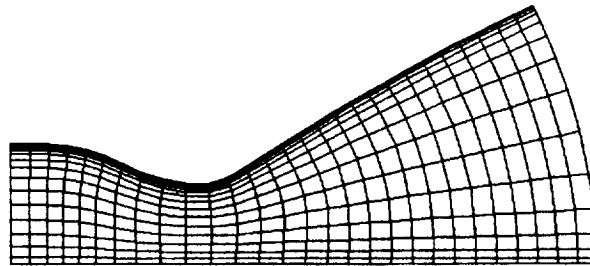> of the code.

Figure 31.—Stretched grid.

In "Relative" stretching, users can specify the size of the first mesh from the bound-
aries by typing in a fraction of the average mesh size at the narrowest point of the
geometry. In this example, the average mesh size is (yp1−yp2)/(Number of meshes) =
(0.670655−0)/15.

> NOTE: Specify all necessary stretching parameter values before selecting the
> "Apply" box; never let the top and bottom box values be set at 0.

5.4.2 Grid Refining

TURBO-GRD can refine grid globally and locally. In this section, the uniform grid shown in figure 32 will be refined.
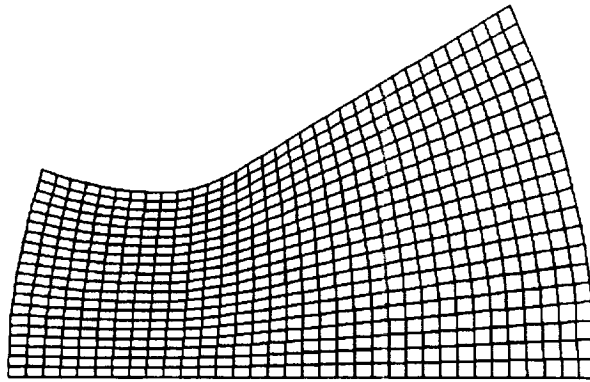

Figure 32.—Uniform grid to be refined.

USER: Return to the HOME menu and select MODIFY GRID (Direct). Then select "Refine Grid."

TURBO-GRD: Displays the number of points in the menu boxes "i-pnt" and "j-pnt" (see fig. 33)

i-pnt   35 |

j-pnt   21 |

Apply   Return

Figure 33.—Menu for grid refinement.

USER: Increase the number of points in the j-direction from 21 to 42 (fig. 34). Select the "Apply" box to carry out the change.

i-pnt   34 |

j-pnt   42 |

Apply   Return

Information

Block = 1
i = 1
j = 1
# points on edge 1: 42
# points on edge 2: 35
# points on edge 3: 42
# points on edge 4: 35

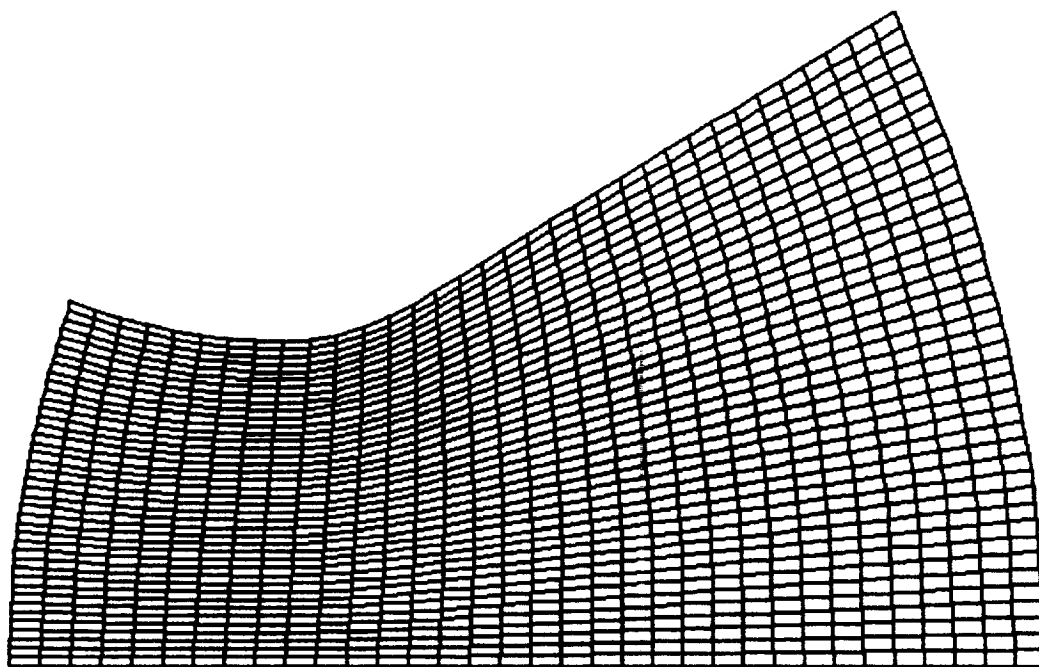Figure 34.—Number of points increased to 42 in j.

Figure 35.—Global grid refinement.

The global grid refinement may change the grid structure, which may require some modification. While refining, one way to keep the grid structure is to refine in subregions as shown in the next section.

How To Refine Subregions of a Block

With TURBO-GRD, the user can choose a subregion of the domain and refine the grid in that region.

USER: From the MODIFY GRID (Direct) menu, select "Choose Active Subregion." Use the LM-key to choose a point (i, j).Then use the MM-key to choose another point (i+m, j+n) to define the subregion.

TURBO-GRD: Highlights the selected subregion

USER: Execute the "Refine Grid" operation as illustrated earlier in 5.4.2.

TURBO-GRD: Refines subregion grid as it refined the entire domain grid

The user can refine as many subregions as needed. Figure 36 shows the resulting grid after two subregion refinements.
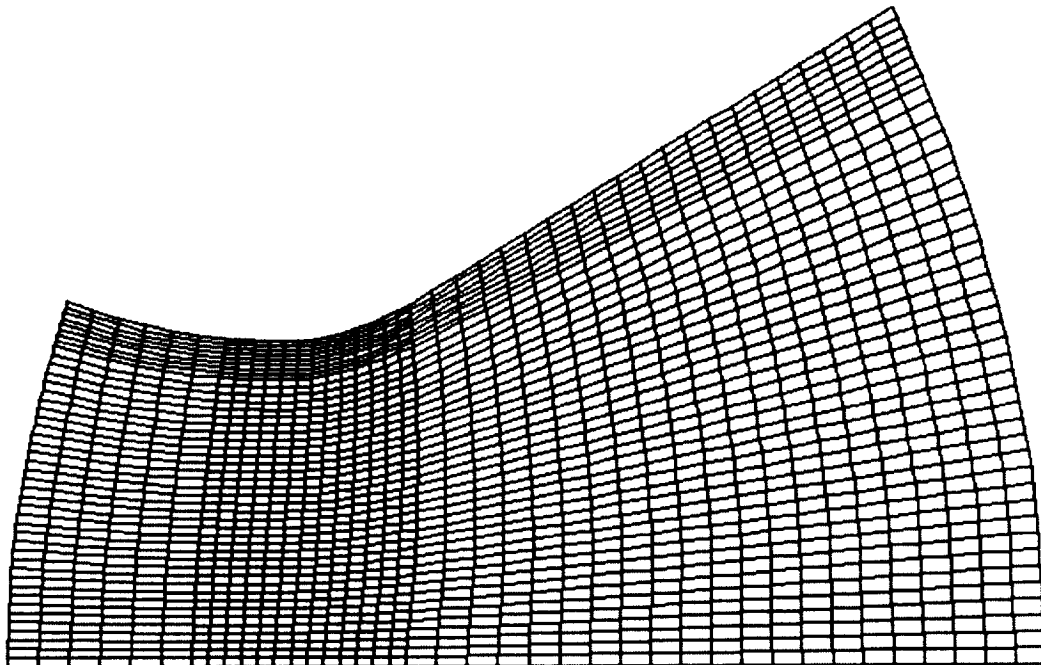
Figure 36.—Grid refinement of interactively selected local subregions.

After subregions are refined, the user should redefine the block for the entire domain and then save the file for either a flow simulation, a later session of TURBO-GRD, or further modifications.

NOTE: Save grid files often so that the TURBO-GRD process can be restarted from that grid when unexpected problems occur.

How To Stretch Grid Using Stretching Functions

TURBO-GRD provides grid stretching by utilizing stretching functions. Geometric stretching and hyperbolic tangent stretching are available. As an illustration, the grid shown in figure 36 is stretched using the submenu "Stretch Grid" from the MODIFY GRID (Direct) menu. Before stretching the grid, the user should choose the active sub-region that includes the entire domain. The stretched new grid is shown in figure 37.
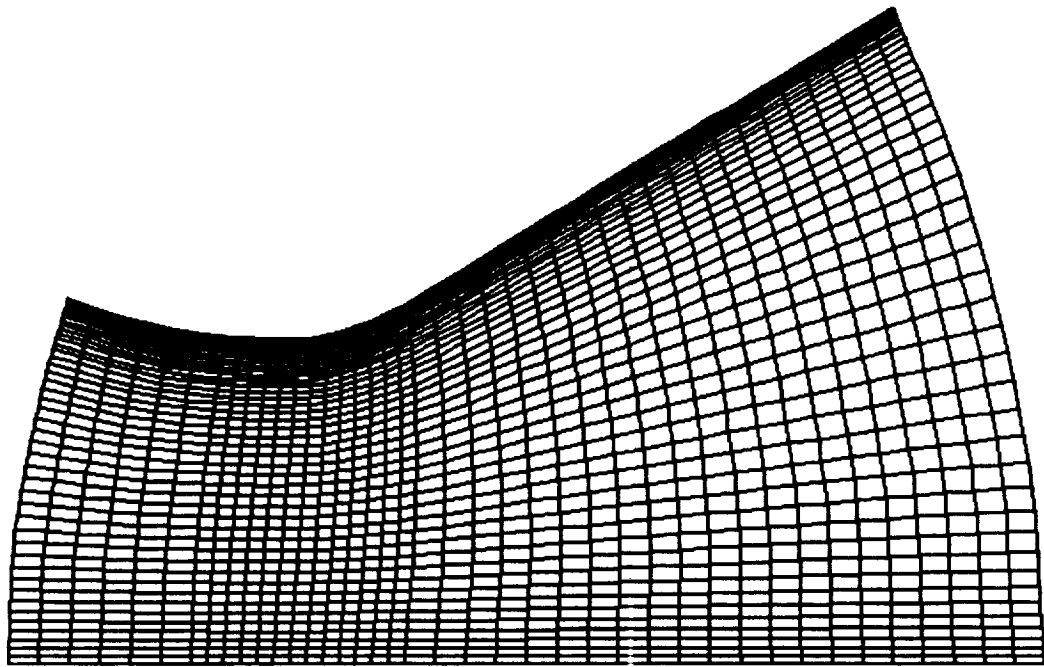
Figure 37.—Refined, stretched grid.

6.0 TURBO-GRD FEATURES THROUGH MULTIBLOCK EXAMPLES

6.1 Example 4: Interactive Domain Decomposition

TURBO-GRD allows the user to perform domain decomposition interactively by utilizing a Bézier curve. The shape of the internal boundary thus formed can be manipulated using the two interior control points of the Bézier curve. A single domain to be decomposed is shown in figure 38.

Figure 38.—Single domain to be decomposed.

USER: From the GENERATE BNDRY/FIELD GRID menu, select "Subdivide Block."
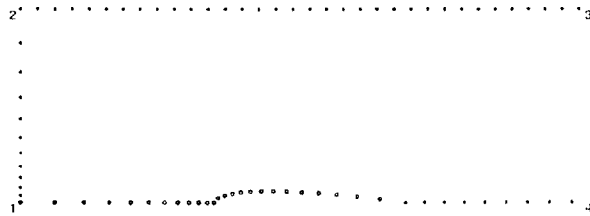Follow the instructions on the monitor to pick two end points of the desired
interior boundary; use the LM-key for end point P1 and the RM-key for end point
P2.

TURBO-GRD: Displays x,y-coordinates of P1 and P2 at the lower left corner of the
monitor and displays a straight dotted line that connects these two points.
It initially uses 10 default points, as shown in figure 39.

P1

x = 3.38536E + 01
y = 0.00000E + 00

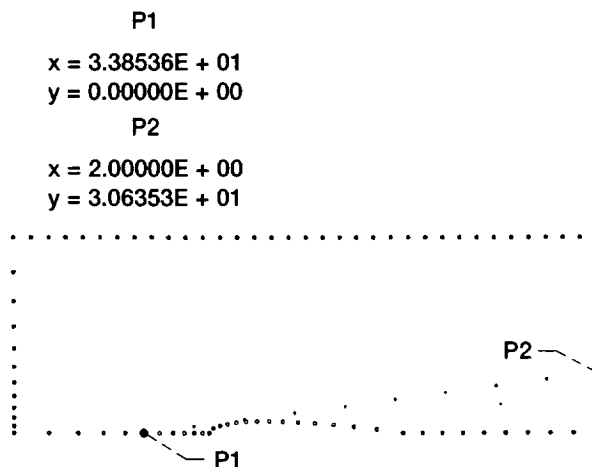P2

x = 2.00000E + 00
y = 3.06353E + 01

Figure 39.—Selection of end points of internal block
boundary.

USER: Press the RM-key to finalize the selection of the two end points.

TURBO-GRD: Displays the initial Bézier curve with two control points in the interior, as
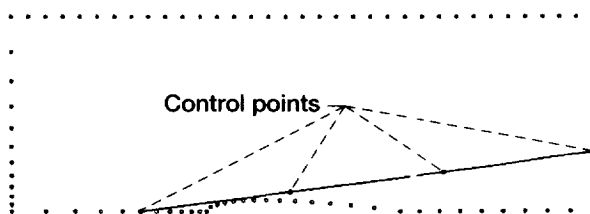shown in figure 40

Figure 40.—Initial Bézier curve and control points.

USER: Move the cursor to an interior control point and grab it by holding down the
     LM-key. Operate on one interior CP at a time. Continue to hold down the mouse
     key and move the control point until the interior boundary changes to the desired
     shape.

TURBO-GRD: Changes the shape of the new internal block boundary in real time. Fig-
     ure 41 shows the resulting interior boundary curve and the interior con-
     trol points at new locations.
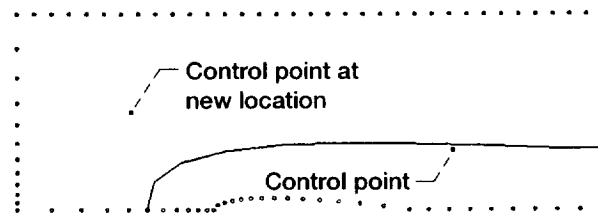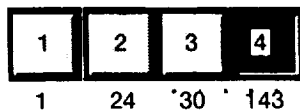

Figure 41.—Shaping internal boundary using control
points.

USER: From the menu GENERATE BNDRY/FIELD GRID, select corner points
     (fig. 14) and discretize edges before generating the grid.

TURBO-GRD: Shows an interim result (fig. 42) where one-block boundary grid with the
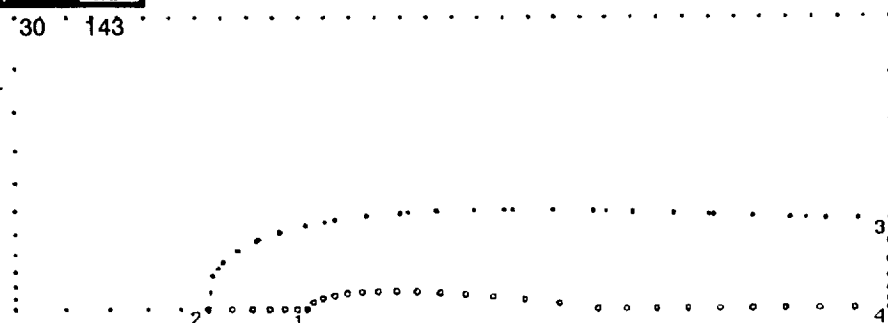   corners defined is ready to generate field grid

Figure 42.—Completion of one-block boundary grid.

USER: Select "Switch Blocks" and repeat the proces:. Use the "Copy Subcurve Distribution" and "Apply Subcurve Distribution" features if desired, as illustrated in example 5 in the next section.

6.2 Example 5: Multiblock Boundary and Field Grid Generation

In this example, the multiblock capability of TURBO-GRD is illustrated by using a three-block case. All the TURBO-GRD features explained in examples 1 to 4 are valid for the boundary and field grid generation in each block of example 5. Therefore, detailed explanations of TURBO-GRD features dealt with in previous examples will not be repeated here; instead, new features needed in the multiblock operation will be emphasized.

Input files of example 5 are file5.geo and file5.par, which are presented in tables 15 and 16, respectively.

NOTE: file5.geo in table XV is three-block data with each block having four data points.

TABLE XV.—INITIAL THREE-BLOCK
GEOMETRY DATA (file5.geo)

		3	Three-block data
		4	4 Points in block 1
		4	4 Points in block 2
		4	4 Points in block 3
1.00	0.00		First (x,y) data of block 1
1.00	1.00		
5.00	1.00		
5.00	0.00		Fourth (x,y) data of block 1
0.00	1.00		First block 2 data
0.00	3.00		
1.00	3.00		
1.00	1.00		Last block 2 data
1.00	1.00		First block 3 data
1.00	3.00		
5.00	3.00		
5.00	1.00		Last block 3 data

In table XVI, Data_typ_in = 1 in file5.par, which indicates that file5.geo contains boundary data. For the boundary data input file, the parameters imin, imax, jmin, jmax, icnmax, and jcnmax do not have any meaning at this point as they are related to the grid and control net. Nevertheless, the last three lines of data in table XVI are needed to run TURBO-GRD. The icnmax and jcnmax values will be the number of control points to be used when grid is generated. The user can change these values interactively after grid is generated.

TABLE XVI.—INITIAL THREE-BLOCK PARAMETER FILE (file5.par)

file5.par					
Data_typ_in	Data_typ_out				
1	2				
#blocks					
3					
imin	jmin	imax	jmax	icnmax	jcnmax
1	1	1	0	4	4
1	1	1	0	4	5
1	1	1	0	4	5

6.2.1 Displaying Input Data in file5.geo

USER: Type in "turbogrd."

TURBO-GRD: Please enter input file name!

USER: file5

TURBO-GRD: Is there a PLOT3D solution file to read? If so, what is the format of the file?
  (0) No; (1) Formatted; (2) Unformatted; (3) Binary

USER: 0

TURBO-GRD: Reads in file5.geo and file5.par and displays the geometry data with block 1 data highlighted in red (on monitor) as shown in figure 43
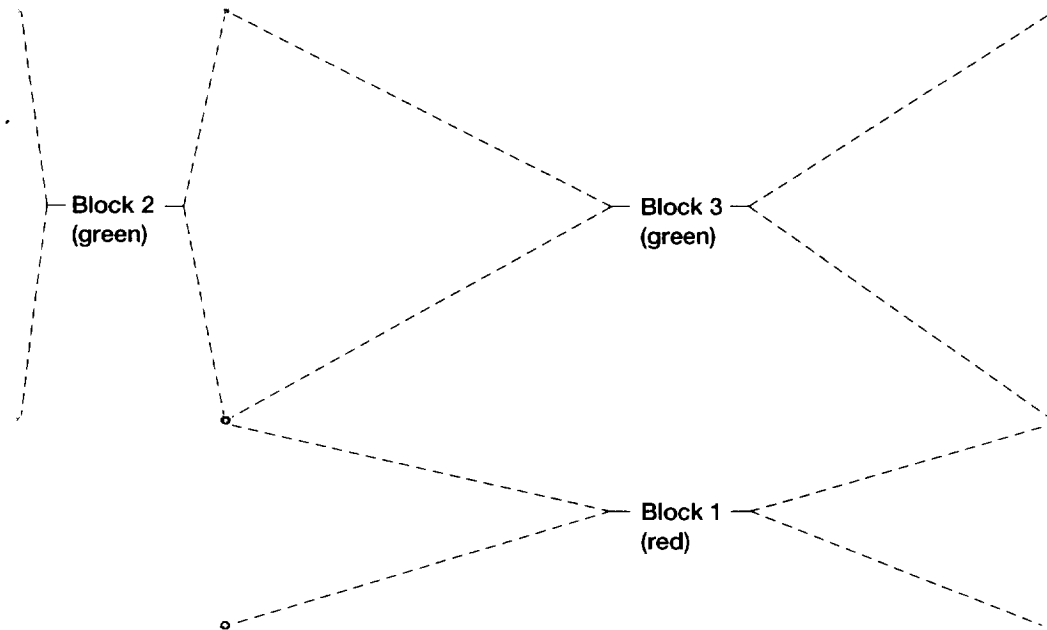

Figure 43.—Three-block geometry data from file5. geo.

6.2.2 Choosing Block To Work On

USER: Work on block 1 or work on other blocks by selecting "Switch Blocks" from the GENERATE BNDRY/FIELD GRID menu. Wait until the desired block is highlighted in red.

The boundary grid can be generated for each block as illustrated in example 1 in section 5.2. Instead of repeating the process followed for example 1, we present three-block boundary grid in figure 44. On the workstation monitor, the boundary grid points of blocks 1 and 2 are green and of block 3 are red. The highlighted red points indicate that the user is currently working on block 3. Block 3 has stretched boundary grid whereas blocks 1 and 2 have uniform boundary grids.

NOTE: Nonmatching grids along the common interior block boundaries are produced by the code at this point.

USER: From the GENERATE BNDRY/FIELD GRID menu, select "Construct Initial CN and GRID" and then from the pulldown menu, choose "Generate Initial Grid for all Blocks at Once."
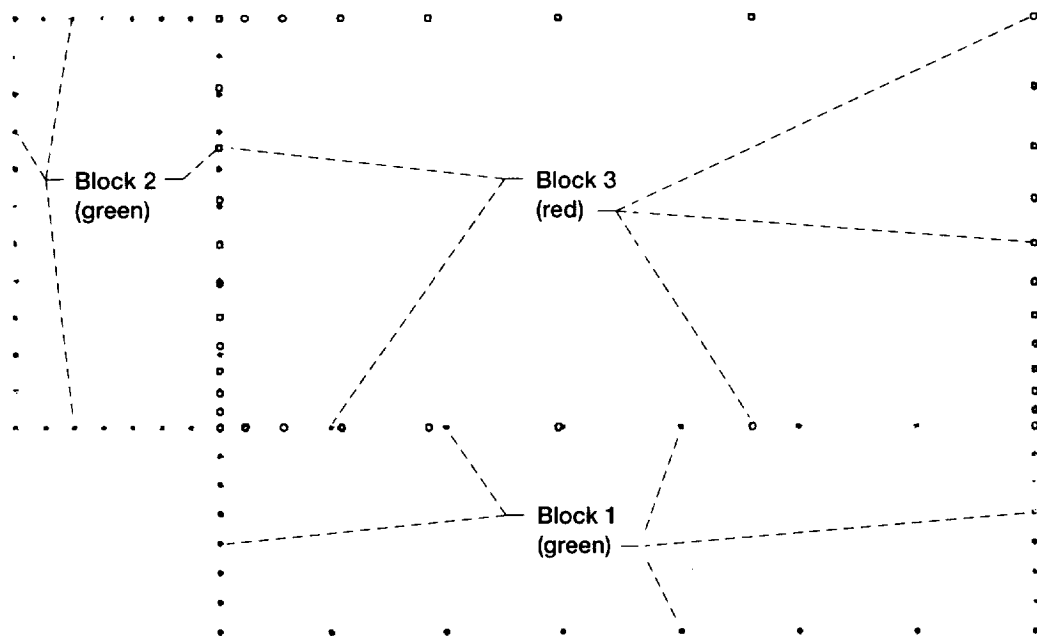
Figure 44.—Three blocked boundary grids with nonmatching interior boundary grid distribution.

6.2.3 Generating Blocked Grid

USER: Set corners of blocks 1, 2, and 3 before generating grid. From the GENERATE BNDRY/FIELD GRID menu, select "All Blocks" under the pulldown menu "Construct Initial CN and GRID."

TURBO-GRD: Generates grid for all blocks, takes the process to HOME, and displays the three-block grid on the monitor. Initially, block 1 will be displayed with a control net in blue and a highlighted grid.

USER: In the HOME menu, select "Switch Blocks" twice to activate block 3.

TURBO-GRD: Displays the grid shown in figure 45. The grid is C^0 discontinuous across the interior block boundaries.
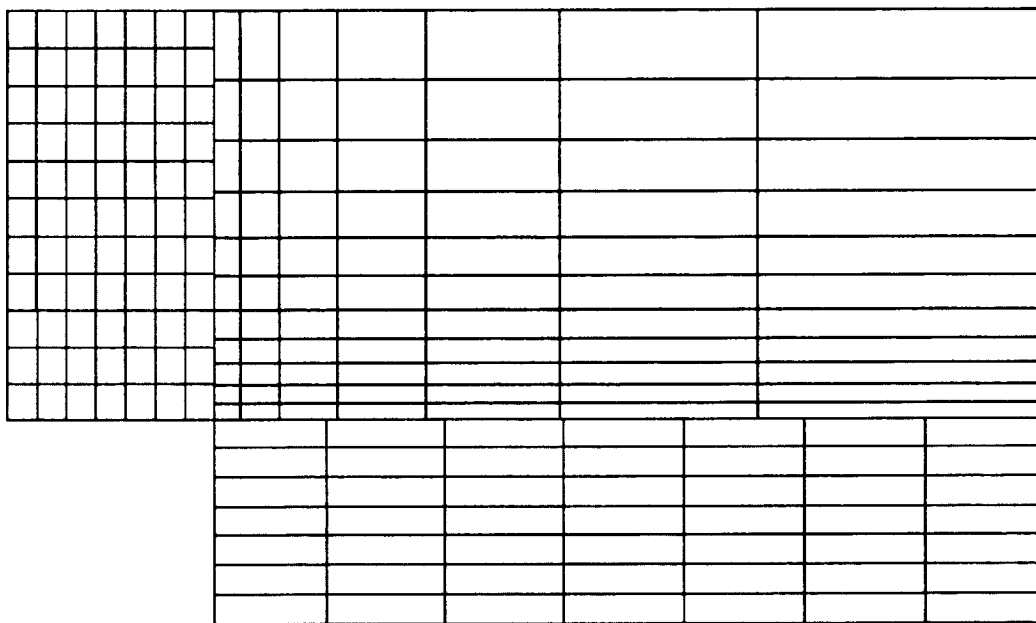

Figure 45.—Discontinuous grid across blocks.

6.2.4 Making Grid C^1 Continuous Across Common Interior Boundaries

How to Generate Grid for One Block at a Time

USER: Generate the boundary grid for one block (e.g., block 1). Set corners for block 1. From the GENERATE BNDRY/FIELD GRID menu, select "Current Block" from the pulldown menu of "Construct Initial CN and GRID."

TURBO-GRD: Generates field grid for block 1 and takes the process to HOME

USER: Examine block 1 grid. At HOME now, select the GENERATE BNDRY/FIELD GRID menu and then the "Switch Blocks" submenu to activate block 2. Generate boundary grid for block 2. Set corners. Then select the "Current Block" in the pulldown menu of "Construct Initial CN and GRID."

TURBO-GRD: Generates grid for block 2 and displays block 1 and block 2 grid at HOME

USER: Repeat the process for block 3.

How To Make Grid Continuous Across Common Interior Boundaries

In these instructions, the user will learn to use the features "Copy Subcurve Distribution" and "Apply Subcurve."

USER: From the GENERATE BNDRY/FIELD GRID menu, select "Choose Subcurve" and choose the interior boundary of block 3 that is common to block 1. Try to remember which mouse keys were used to choose each end point.

TURBO-GRD: Highlights in red the chosen boundary grid points

USER: Select "Copy Subcurve Distribution" and ther "Switch Blocks."

TURBO-GRD: Highlights block 1, which becomes active

USER: Choose the interior boundary of block 1 that is common to the copied block 3 boundary by using the option "Choose Subcurve."

NOTE: Use the LM-key and the MM-key for the same end points in blocks 3 and 1 when choosing the subcurve. If the wrong mouse keys are used, simply select the subcurve again by clicking the LM- ar d MM-keys for different end points.

TURBO-GRD: Highlights in red the boundary grid points on the chosen subcurve of block 1

USER: Choose "Apply Subcurve Distribution."

TURBO-GRD: Copies the distribution of block 3 boundary onto the common boundary
of block 1. This process matches exactly the grid points of block 1 with
those of block 3.

Repeat the "Copy Subcurve Distribution" and "Apply Subcurve Distribution" processes
for blocks 2 and 3. The modified boundary grid is shown in figure 46.

As mentioned in example 1 (section 5.2), the user's responsibility is to confirm the
same number of grid points on the opposing boundaries and to set corners before generating field grids by selecting the submenu "Construct Initial CN and GRID."
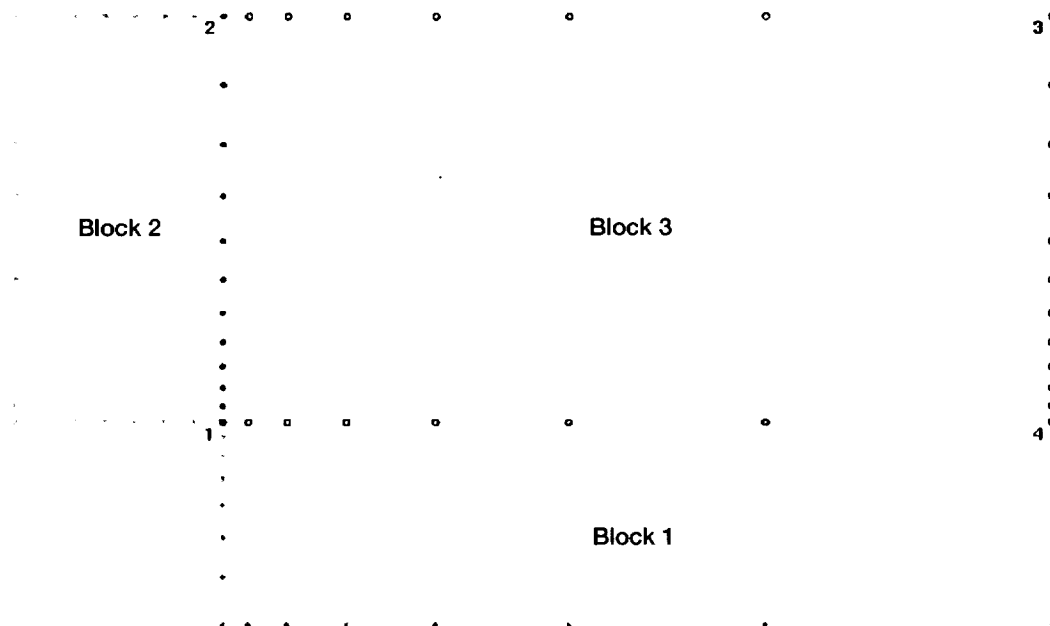

Figure 46.—Matched grid on common interior boundary.

The modified grid generated from this new boundary grid is displayed in figure 47.
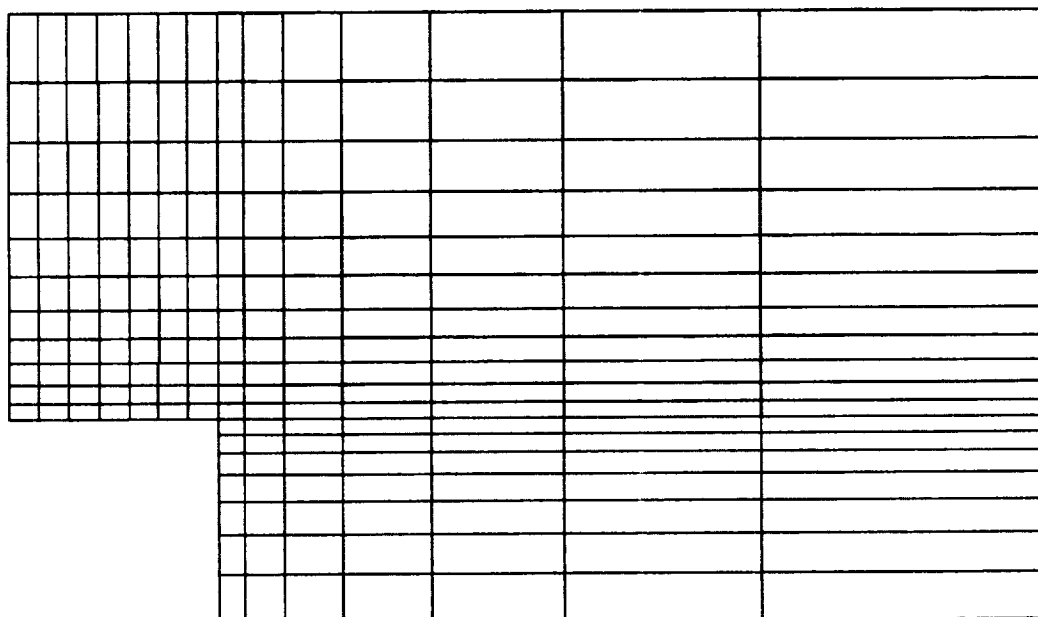
Figure 47.—Continuous grid lines across blocks.

The user can stretch the grid further (fig. 48) by using the submenus "Stretch CN" and "Recalculate Grid" from the MODIFY GRID (Through CN) menu. To ensure continuous grid across the common interior block boundaries, the submenus "Copy Subcurve Distribution" and "Apply Subcurve Distribution" need to be used from the GENERATE BNDRY/FIELD GRID menu.
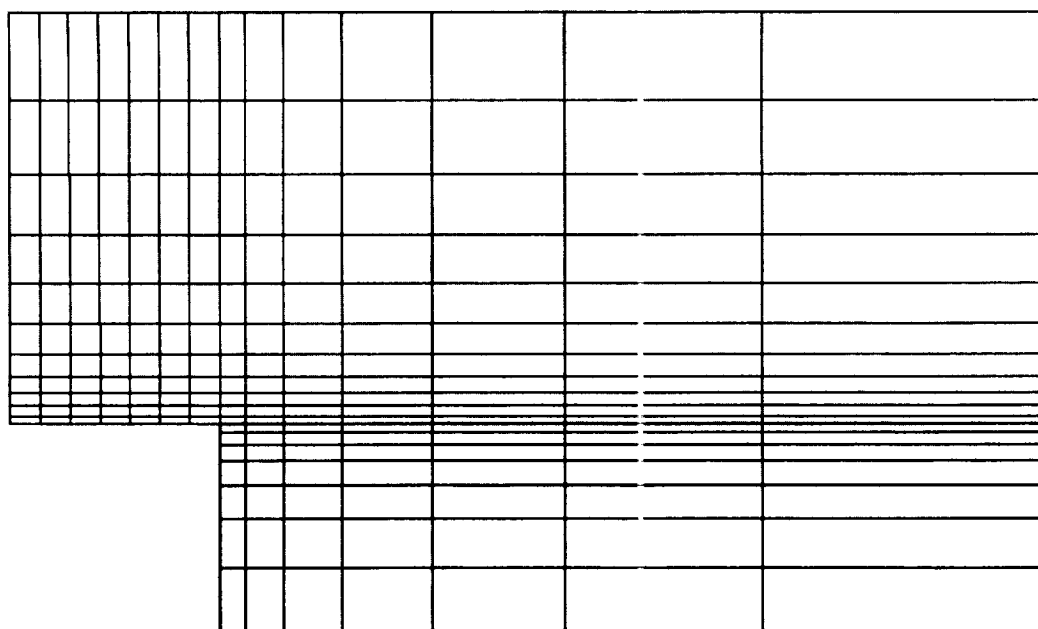

Figure 48.—Further stretched grid.

How To Make Uniform Distributions on the Right Side Edges of Blocks 1 and 3
To Reduce Mesh Aspect Ratios

USER: At HOME, choose the menu GENERATE BNDRY/FIELD GRID.

TURBO-GRD: Displays the boundary grid

USER: In block 1, choose the boundary on the right by selecting "Choose Subcurve."
Then select "Discretize Subcurve" and set unif to 10. (See fig. 11 in Sec. 5.2.1.)

TURBO-GRD: Gives a uniform distribution on the subcurve

USER: Press RM-key to return to the GENERATE BNDRY/FIELD GRID menu. Using
the "Switch Blocks" submenu, activate block 3. Select the right side boundary
using "Choose Subcurve." Select the "Discretize Subcurve" submenu. The value
of unif does not need to be changed because unif is already 10.

TURBO-GRD: Gives a uniform distribution on the subcurve

USER: Press the RM-key to return to the menu. Choose "All Blocks" from the pulldown
menu of "Construct Initial CN and GRID" in GENERATE BNDRY/FIELD
GRID.

NOTE: The corners do not need to be reset because they were established previously.

To show the modified grid well, remove the control net from the display by choosing
the "Toggle Display of CN" from the HOME menu.

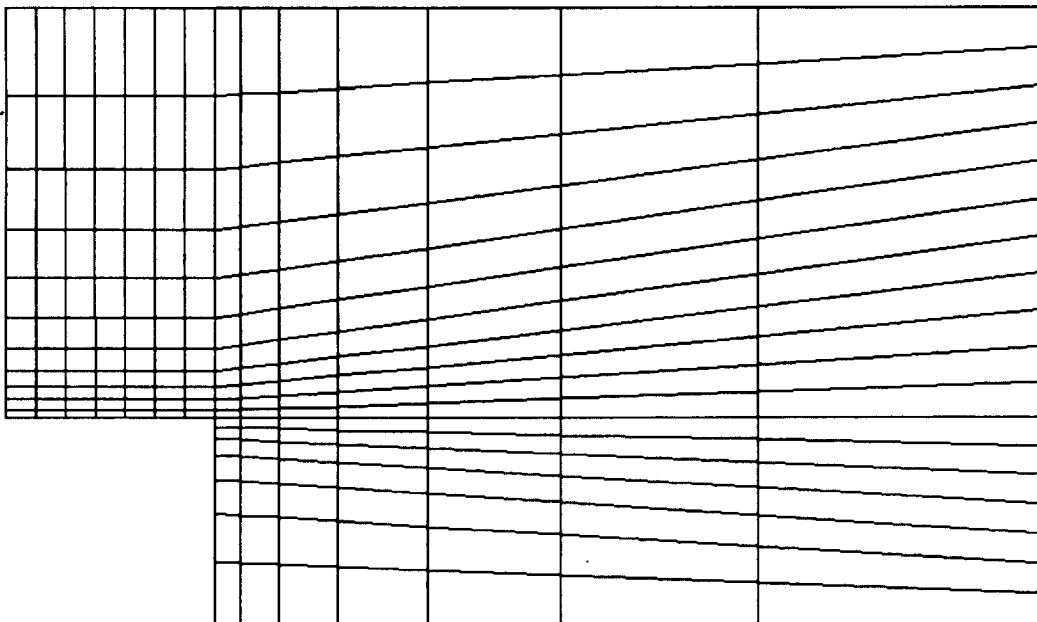TURBO-GRD: Displays the modified grid shown in figure 49.

Figure 49.—Meshes with moderate aspect ratios.

6.3 Example 6: C–H Multiblock Grid for Airfoil

A C–H multiblock grid is generated for an airfoil. The domain boundary for grid generation includes the top half of the airfoil with no turning, as shown in figure 50.

2•   3•

1•   °°° ° °°°°°° □ °   □   °   □   •   4•
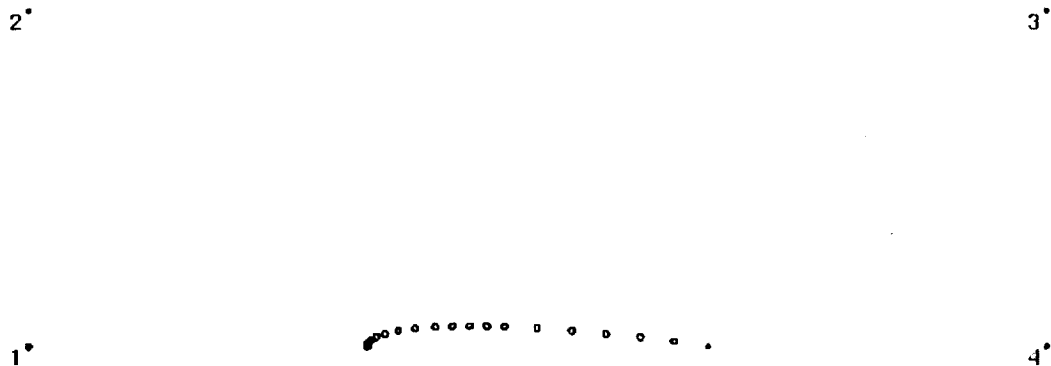
Figure 50.—Flow domain for example 6.

All the TURBO-GRD features for this application have been explained in previous examples. The steps to be taken by the user to generate the C–H-blocked grids are presented below.

1. Place grid points along airfoil surface with proper clustering near leading and trailing edges and with smooth grid spacing.
2. Place grid points along the upstream symmetry line and the downstream wake line with proper clustering and smoothing.
3. Place grid points on other boundaries with enough density and spacing.

NOTE: So far we have a single block. Now the user will begin to divide it into three blocks.

4. First, subdivide the block by creating the C-grid around the airfoil. Choose a grid point on the upstream symmetry line and then a grid point on the downstream outflow line. Modify the Bézier curve shape until it follows smoothly the shape of the airfoil, as shown by the curve between points 1 and 2 in figure 51.
5. Select corners for the C-grid block.

NOTE: Now the user has two blocks.

6. Discretize and distribute grid points along the newly created curve.
7. Switch to the upper block and perform the same discretization of the subcurve adjoining the C-grid. Copy the distribution from the C-grid boundary.

8. Divide the upper block at the "knee" of the C-grid boundary between points 1 and 2, as shown in figure 51. Select the corners of the two newly created blocks.

9. Go through all the blocks and discretize and redistribute grid points. At the boundaries between blocks, ensure that the number of grid points and the grid point distribution match. Use the submenus "Copy Subcurve Distribution" and "Apply Subcurve Distribution" to ensure that grid points are overlapped, if so desired. The resulting boundary grid is shown in figure 51.
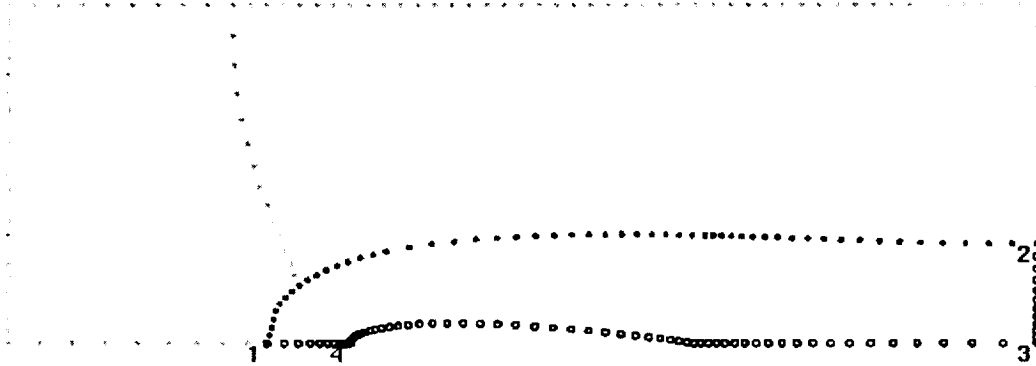

Figure 51.—Blocked domain obtained using Bézier curves.

10. Generate the field grids and control nets for all blocks.

11. Apply grid modifications as desired through direct modification of the grid (e.g., elliptic smoothing) or through the control net. The resulting grid is shown in figure 52 .
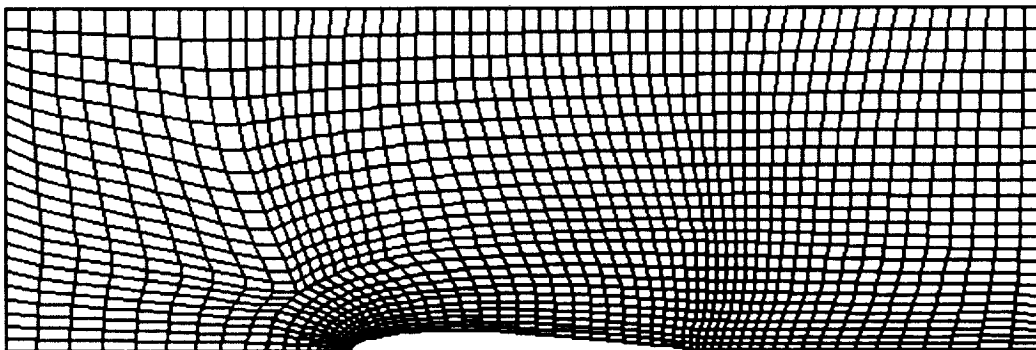
Figure 52.—C-H multiblock grid.

6.4 Example 7: Grid for Airfoil With Ice Accretion

A blocked grid is generated for an NACA0012 airfoil with ice accretion. For this particular ice shape, the grid quality is controlled by the shape of the interior block boundaries. Because TURBO-GRD uses a Bézier curve for the interior boundary, its shape can be interactively controlled by manipulating the two interior control points of the Bézier curve. In figure 53, for example, the interior block boundary that connects points 1 and 2 is a Bézier curve. Detailed steps for interactive domain decomposition are presented in example 4 of section 6.1.
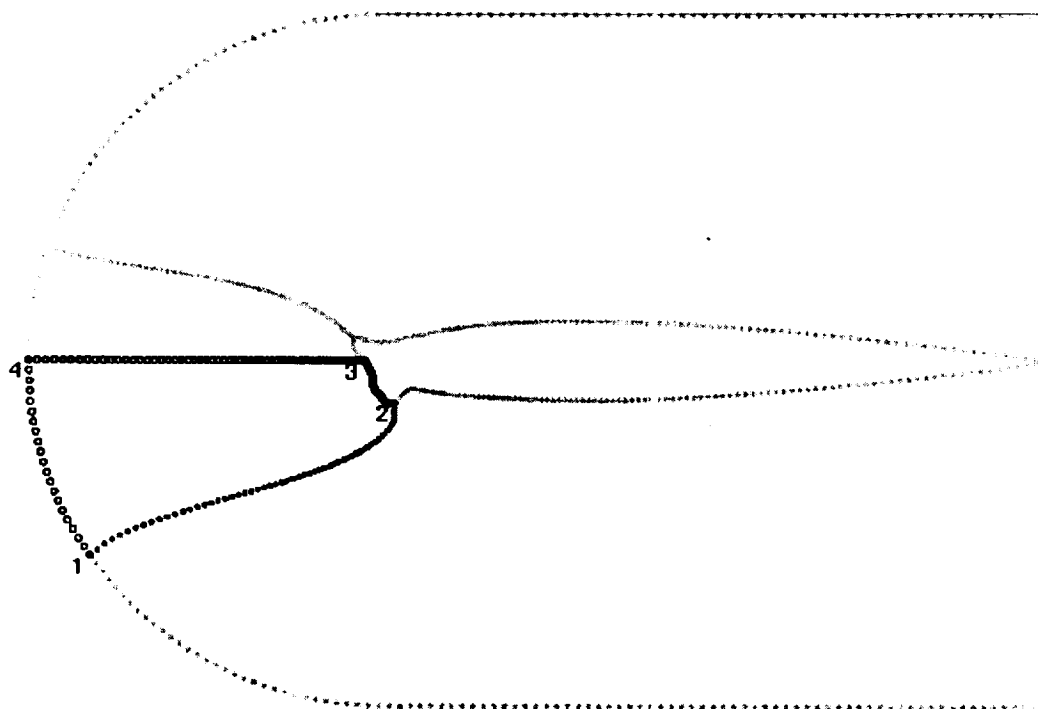
Figure 53.—Blocked domain around NACA0012 airfoil with ice accretion.

A four-block grid generated from the boundary grid of figure 53 is stretched and displayed in figure 54. Multiblock field grid generation from the blocked domain is discussed in detail in example 5 of section 6.2, and the steps for grid stretching are presented in example 3 of section 5.4.

Figure 54.—Grid around airfoil with ice accretion.

An enlarged view of the grid around the leading edge is shown in figure 55.

Figure 55.—Grid around leading edge.

7.0 SYSTEMATIC ICE SHAPE SMOOTHING

A given geometry such as ice shapes may have many sharp corners and high curvature segments. Curve discretization and eventual field grid generation on such geometry are very difficult and time consuming. TURBO-GRD offers a feature by which users can perform geometry smoothing in a controlled manner by using control points. Follow these steps to use this feature:
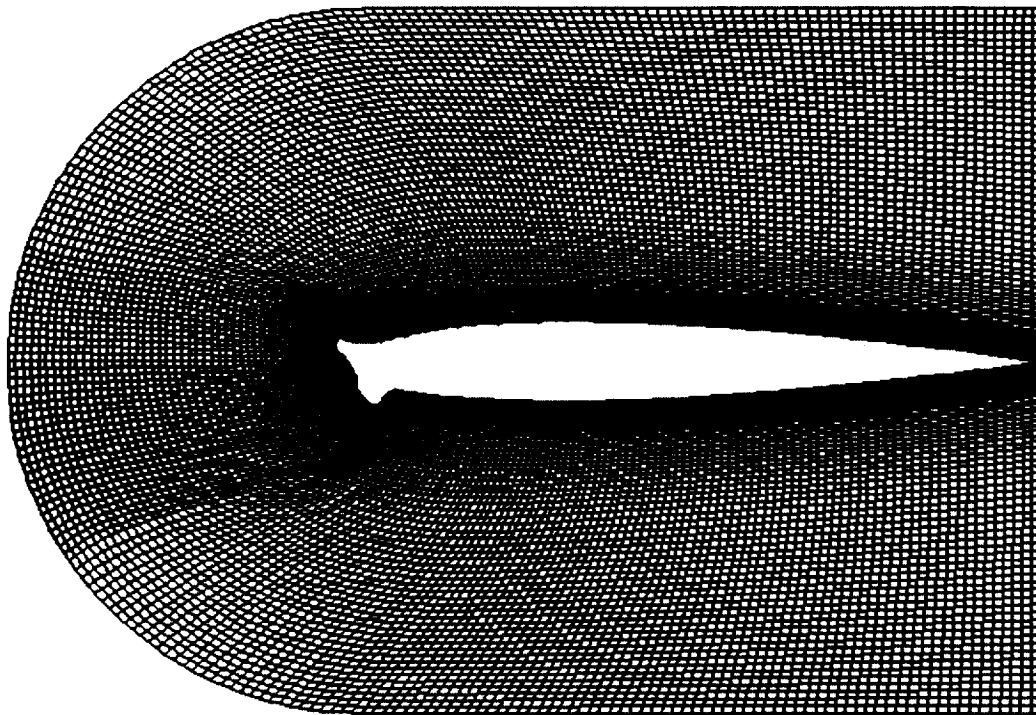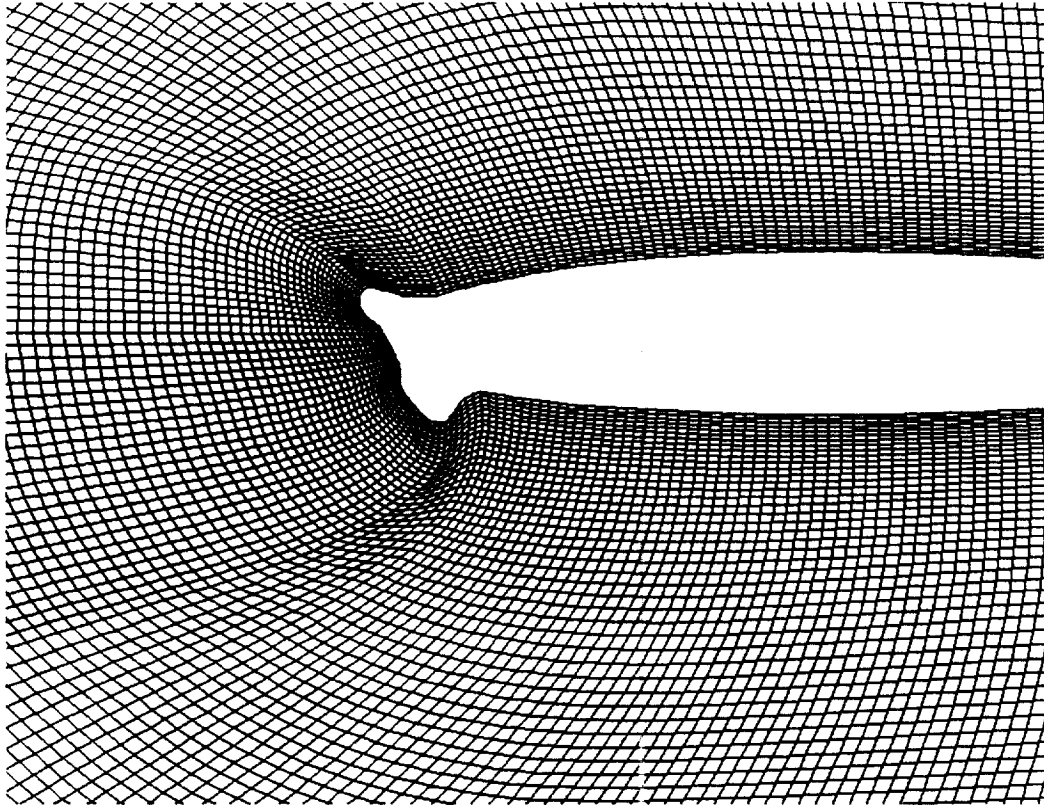
1. From the GENERATE BNDRY/FIELD GRID menu, select "Free Form Control."

TABLE XVII.—FREE FORM CONTROL MENU

Free Form Control
Return
Choose FF Segment
Change FF Boundary

2. From the "Free Form Control" submenu, select "Choose FF Segment." Follow the instructions in the information area to select the curve segment for the free form (i.e., smoothing) operation. The selected segment will be highlighted in red. Press the RM-key to return to the "Free Form Control" submenu.

3. From the "Free Form Control" submenu, select "Change FF Boundary." The dialogue boxes and information area appear in the menu box. The number of data points on the selected segment is presented in the information area. A default number of 9 control points is displayed in the "nFFCP" box. A very close approximation of the smoothed curve can be obtained when the value of nFFCP is equal to the number of data points on the FF segment.
As the user reduces the value in the nFFCP box, the new curve becomes smoother as its shape further changes from the original.

4. The user can experiment with different levels of smoothing by changing the nFFCP value and by selecting "OK." (No permanent change has occurred yet because the original curve geometry is still displayed in red and the new smoothed curve in blue.) The user can repeat this test cycle until the desired level of smoothing is achieved.

NOTE: To aid the user for close inspection of the smoothing, use "Z-In" to zoom to a particular area and "Z-Out" to see the whole graphical object.

5. When the desired nFFCP value is found, choose "Apply" to replace the original curve segment with the new smoothed segment.

6. Use the RM-key to select "Return" to go back to the "Free Form Control" submenu. Then, again select "Return" to go to the GENERATE BNDRY/FIELD GRID menu and choose "Save Boundary Grid." The new smoothed curve will be saved in files called bound0.geo and bound0.par.

7. If rediscretization is desired, use the "Discretize Subcurve" submenu discussed in Example 1 (sec. 5.2).

.

8.0 NOTES TO USERS

1. The code runs on an SGI workstation with a GL graphics library.

2. The code has the following limits:

   Maximum number of blocks: 9
   Grid size of each block: 300 × 300
   Boundary data per block: 1400

3. In section 5.4.2, Grid Refining, numbers typed very rapidly into numeric fields (see fig. 33) may not be read in correctly.

4. Block boundary data (Data_typ_in = 1) are line data. As shown in table I in section 3.1.1 and table XIII in section 5.2, the coordinate (x,y) of the last data point needs to be specified even though it is identical to that of the first point.

5. Expose events sometimes leave black holes instead of redisplaying the graphics in the window. If this happens, users need to select a menu entry and return to the desired interactive point to recover the graphics object and window.

6. From the "Discretize Subcurve" feature from the GENERATE BNDRY/FIELD GRID menu, the following terms are defined:

"unif" is a parameter to obtain a uniform distribution of grid. For example, unif = 0 keeps the existing distribution, unif = 10 changes the distribution to uniform, and any value between 0 and 10 results in an intermediate distribution.

"crv" is a parameter to put more points where the curvature is higher. This is not well tested yet.

"pnts" indicates the number of points the user wants to have on the selected subcurve.

NOTE: If the initial subcurve consists of only two end points, the unif does not work. The user should first add points as desired by choosing "Apply" and then "Return;" then select "Discretize Subcurve" again and set the desired value of unif. There is no undo feature for unif; instead, the user can employ the "stretch" feature for redistribution.

7. In the GENERATE BNDRY/FIELD GRID menu, it is now the user's responsibility to choose "Select Corners" for the topologically rectangular block and to assure that the same number of grid points exist on opposing boundaries before selecting the "Construct Initial CN and GRID."

8. The numeric keypad does not work for making numeric entries in dialogue boxes.

9. In figure 13 in section 5.2.1, the four push buttons displayed in the "Select Corners" mode may cover up some of the corner points to be selected. If this situation exists, use the "Gradual Zoom and Move" feature in the MODIFY VIEW menu to translate the graphical object to the desired location by moving the mouse while the LM-key is depressed.

9.0 REFERENCES

1. K.D. Lee; and J.M. Loellbach: Geometry-Adaptive Surface Grid Generation Using Parametric Projection. J. Aircraft, vol. 26, Feb. 1989, pp. 162–167.
2. T.L. Henderson; Y.K. Choo; and K.D. Lee: Interactive Solution-Adaptive Grid Generation Procedure. NASA TM–105432, 1992.
3. P.R. Eiseman: A Control Point Form of Algebraic Grid Generation. Int. J. Num. Methods in Fluids, vol. 8, Oct. 1988, pp. 1165–1181.
4. P.R. Eiseman; Y.K. Choo; and R.E. Smith: Algebraic Grid Generation With Control Points. Finite Elements in Fluids, Edited by T.J. Chung, Hemisphere Publishing, vol. 8, pp. 97–116, 1992.

REPORT DOCUMENTATION PAGE

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY (Leave blank)	2. REPORT DATE October 1998	3. REPORT TYPE AND DATES COVERED Technical Memorandum	
4. TITLE AND SUBTITLE User Manual for Beta Version of TURBO-GRD: 2A Software System for Interactive Two-Dimensional Boundary/Field Grid Generation, Modification, and Refinement			**5. FUNDING NUMBERS** WU–548–20–23–00
6. AUTHOR(S) Yung K. Choo, John W. Slater, Todd L. Henderson, Colin S. Bidwell, Donald C. Braun, and Joongkee Chung			
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135–3191			**8. PERFORMING ORGANIZATION REPORT NUMBER** E–11082
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546–0001			**10. SPONSORING/MONITORING AGENCY REPORT NUMBER** NASA TM—1998-206631 ICOMP–98–02

11. SUPPLEMENTARY NOTES

Yung K. Choo, John W. Slater, Colin S. Bidwell, and Donald C. Braun, NASA Lewis Research Center, Cleveland, Ohio 44135; and Todd L. Henderson, Raytheon Systems Company, Waco, Texas 76715-4580, and Joongkee Chung, Ohio Aerospace Institute, Cleveland, Ohio 44142. Responsible person, Yung K. Choo, organization code 5840, (216) 433–5868.

12a. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified - Unlimited Subject Categories: 02, 34, and 61 Distribution: Nonstandard This publication is available from the NASA Center for AeroSpace Information, (301) 621–0390.	**12b. DISTRIBUTION CODE**

13. ABSTRACT (Maximum 200 words)

TURBO-GRD is a software system for interactive two-dimensional boundary/field grid generation, modification, and refinement. Its features allow users to explicitly control grid quality locally and globally. The grid control can be achieved interactively by using control points that the user picks and moves on the workstation monitor or by direct stretching and refining. The techniques used in the code are the control point form of algebraic grid generation, a damped cubic spline for edge meshing, and parametric mapping between physical and computational domains. It also performs elliptic grid smoothing and free-form boundary control for boundary geometry manipulation. Internal block boundaries are constructed and shaped by using a Bézier curve. Because TURBO-GRD is a highly interactive code, users can read in an initial solution, display its solution contour in the background of the grid and control net, and exercise grid modification using the solution contour as a guide. This process can be called an interactive solution-adaptive grid generation.

14. SUBJECT TERMS Grid generation; Computational fluid dynamics; Computer software system			**15. NUMBER OF PAGES** 67
			16. PRICE CODE A04
17. SECURITY CLASSIFICATION OF REPORT Unclassified	**18. SECURITY CLASSIFICATION OF THIS PAGE** Unclassified	**19. SECURITY CLASSIFICATION OF ABSTRACT** Unclassified	**20. LIMITATION OF ABSTRACT**

NSN 7540-01-280-5500